



Radica Electra[®] E6

User Manual

Table of Contents

1. Concepts and What's New.....	6
1.1 Concepts.....	6
1.2 What's New in E6	7
2. Getting Started	8
2.1 Starting Electra	8
2.2 Page Scale.....	8
2.3 Page Measurement Units.....	9
2.4 Stencils and Drawing Area.....	10
2.5 Connection Points	11
2.6 Positioning Text.....	12
2.7 Selecting Sub Shapes in a Group	12
3. Document Options.....	13
3.1 Popup Reference Window On Drop.....	13
3.2 Show Symbol Description on Drop.....	14
3.3 Show Pin Names on Drop.....	14
3.4 How Page Prefix Works	14
3.5 Enable Page Prefix on References.....	15
3.6 Enable Page Prefix on Wires.....	15
3.7 Page Prefix Separator for Wires	15
3.8 Location Format	16
3.9 Location Zone Options	16
3.10 Saving Your Own Document Options	17
4. Symbols and Components	18
4.1 One Intelligent Symbol.....	18
4.2 Placing and Duplicating Symbols.....	18
4.3 Intelligent Automatic Numbering	19
4.4 Editing Symbol Reference	19
4.5 The Reference Window	20
4.6 Displaying Pin Names	20
4.7 Automatic Pin Names.....	21
4.8 Assigning Pin Names.....	21
4.9 Understanding Symbols and Components.....	22

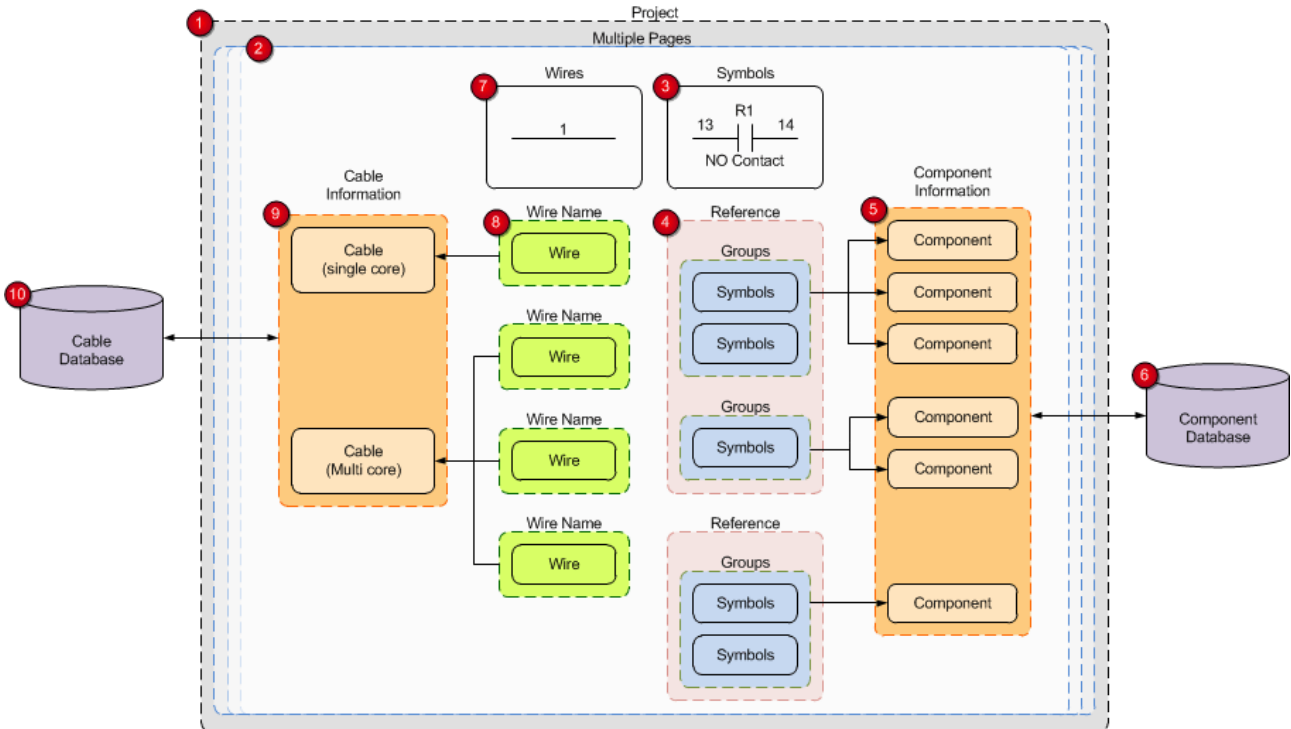
4.10	Assigning Components	23
4.11	Managing Components.....	24
4.12	Selecting a Component.....	26
4.13	The Component Database	28
4.14	The AutoLocation Shape.....	29
5.	Wires and Cables	31
5.1	A Variety of wires	31
5.2	Difference Between A Line And A Wire	31
5.3	Connecting and Manipulating Wires.....	31
5.4	Editing Wire Name	32
5.5	The Wire Window.....	33
5.6	Automatic Wire Status Display.....	33
5.7	Automatic Wire Name Detection.....	34
5.8	Wire Name Propagation.....	34
5.9	Automatic Wire Connection.....	35
5.10	Wire links	36
5.11	Assigning Cables.....	37
5.12	Managing Cables.....	39
5.13	Selecting A Cable	44
5.14	Arranging Cable Cores	45
5.15	The Cable Database	45
5.16	The Cable Tag Shape.....	46
6.	Re-Using Circuits.....	47
6.1	Intelligent Copy and Paste.....	47
6.2	Prefab Circuits	49
6.3	Creating and Using Prefab Circuits.....	50
6.4	Sharing Prefab Circuits	50
6.5	Moving and Duplicating Prefab Circuits.....	50
7.	Intelligent Terminals.....	52
7.1	Terminal Types	52
7.2	Terminal Status and Wire Detection	52
7.3	Generating Terminal Listing	53
7.4	Using Terminal Blocks	54
8.	Panel Layout	57

8.1	The Panel Layout Process	57
8.2	Preparing a Page for Panel Layout	57
8.3	Inserting Plates, Cable Ducts And Rails	57
8.4	Dimension Shapes	58
8.5	Generating Panel Layout	58
8.6	Using Guides to Place Layout Shapes.....	59
8.7	Using 3D Layout Shapes	60
8.8	How Symbols are Matched to Components	60
8.9	Making Your Own Layout Shapes.....	61
9.	Generating Reports.....	62
9.1	Cross Reference Report.....	62
9.2	Material Reference Report.....	63
9.3	Bills of Materials Report	64
9.4	Cable Reference Report	65
9.5	Connections (From/To) Report	66
10.	The PLC Module Shape	67
10.1	Customizing the PLC Module.....	67
10.2	Creating A Single Module with Input And Output.....	68
11.	The Ladder Zone Shape.....	69
11.1	Using Ladder Zone	69
12.	Creating Your Own Symbol	72
12.1	Symbol Creation Process	72
12.2	The Anatomy of a Symbol.....	72
12.3	The Create Symbol Window	73
12.4	Editing the Symbol Icon	77
13.	Creating Your Own Title Blocks.....	78
13.1	The Anatomy Of A Title Block	78
13.2	Title Block Creation Process	79
13.3	Inserting Title Block Zones.....	79
13.4	Inserting Static Text and Graphics	79
13.5	Inserting Automatic Text	80
13.6	Creating and Inserting User Data	81
13.7	The Create Title Block Window	81
13.8	Using a Custom Title Block	82

13.9	Editing the Title Block Icon	82
13.10	Generating Table of Contents.....	83
14.	Miscellaneous	84
14.1	Stencil Browser	84
14.2	Exporting Stencils.....	85
14.3	Importing Stencils	86
14.4	The Renumber Utility.....	87
14.5	Clean Up Document.....	87
15.	Appendix A: Installation and Activation.....	88
15.1	System Requirements.....	88
15.2	Installing Electra.....	88
15.3	Activating Electra	88
15.4	Enabling Macros	89

1. Concepts and What's New

1.1 Concepts



- (1) Users create a new project by opening the New Electra Drawing template.
- (2) Visio drawings can contain multiple pages and each page can contain one or more wires and symbols.
- (3) Symbols represent real world components and always has a reference name (R1), pin names (13 & 14) and description (NO Contact). Right click on a symbol for more operations, including setting pin names and editing references.
- (4) Symbols with the same reference are organized together, and separated into groups by Electra. Each group can then be assigned to one or more real world component.
- (5) Component information are stored in the drawing file for portability. Files that are shared or sent, retain all their data inside the drawing. Users manage component information in a drawing by clicking on menu Electra | Manage Components.
- (6) The component database stores all component specifications for easy access by users, without the need to type and re-type specifications. Users select components from the database to be used on the drawing and can add, edit or delete additional components into the database.
- (7) Wires represents real world cables and always has a wire name. Wires can use wire links to extend to another location or to another page.
- (8) Each wire can be assigned to a cable, or to a specific core of a multicore cable. Numerous wires on a drawing can be assigned to a multicore cable.

- (9) Cable information are stored in the drawing file for portability and can be easily shared or sent without losing their data. Users manage cable information in a drawing by clicking on menu Electra | Manage Cables.
- (10) The cable database stores all cable specifications for easy access by users, without the need to type and re-type specifications. Users select cables from the database to be used on the drawing and can add, edit or delete additional cables into the database

1.2 What's New in E6

- 4x faster scanning and loading at startup
- Better, smarter and more symbols
- Symbols that displays and automatically assign pin names
- New Component database with categories, with new drag and drop interface
- New Select Component interface with category and search
- New AutoLocation shape that monitors symbols and displays location automatically
- More and better wires that connects automatically
- Wire links that detects names, propagates and shows location automatically
- New Cable database, with selectable filters
- Ability to assign wires to single core or multi core cables with automatic assignment
- Drag and drop entire circuits with Prefab Circuits, easily portable and sharable
- New 3D Layout shapes
- New Cable Reference report and Connections (From/To) report
- New LadderZone shape for even easier ladder diagrams
- Plus many more...

2. Getting Started

2.1 Starting Electra

To start a new Electra drawing:

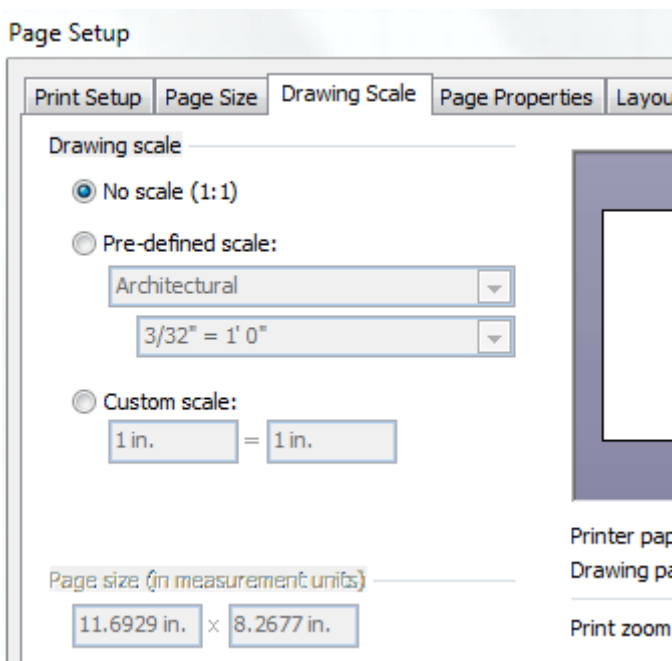
- Click on the Windows Start button
- Select Programs | Radica Software | New Electra Drawing

When Electra is loaded onto Visio, users will see "Electra" on Visio's menu. In Visio 2010, it is under Add-In | Electra. For non-electrical drawings, you can use Visio without Electra. To activate and use Electra for electrical drawings, you must create a new drawing using the New Electra Drawing template.

To open an existing Electra drawing:

- Double click on the drawing file (.vsd) and Visio will automatically load Electra

2.2 Page Scale



In Visio, each and every page can be set to different scales. On the New Electra Drawing 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.

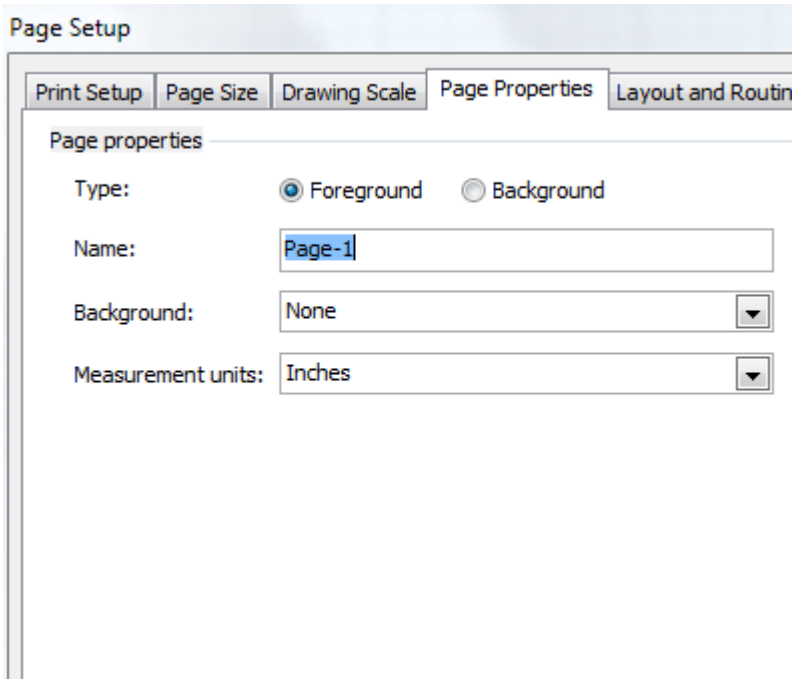
To set page scale:

- Select menu File | Page Setup (Visio 2010: Design tab | Page Setup)
- Click on the Drawing Scale tab
- Click Pre-defined scale or Custom scale
- Select or key in your desired scale

Recommended: A scale of 1:5 for medium sized panels and 1:10 for bigger panels.

See Also: Preparing a Page for Panel Layout

2.3 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 automatically changed to reflect the new measurement units. On the New Electra Drawing template, page measurement units are set to inches and is optimized for schematic drawings. During Panel Layout, you may need to apply a different measurement unit.

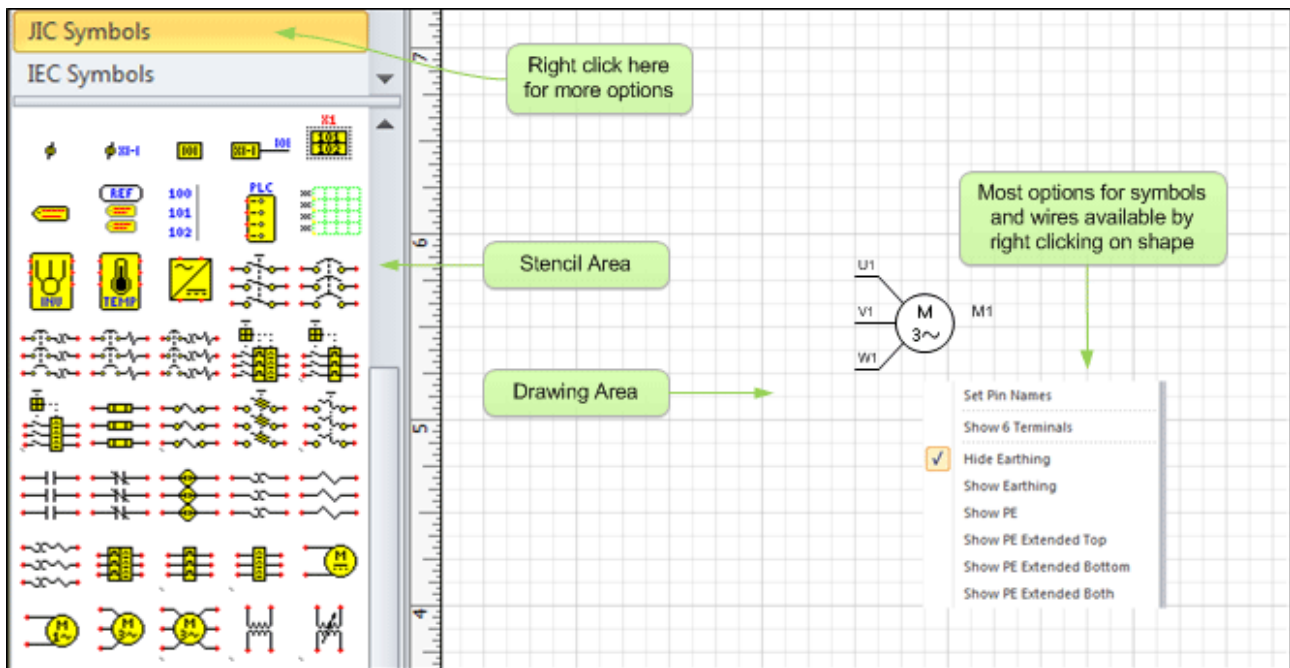
To set the measurement units:

- Select menu File | Page Setup (Visio 2010: Design tab | Page Setup)
- Click on the Page Properties tab
- Click Measurement units and select the desired measurement units

Recommended: Set page measurement units to millimeters during panel layout.

See Also: Preparing a Page for Panel Layout

2.4 Stencils and Drawing Area

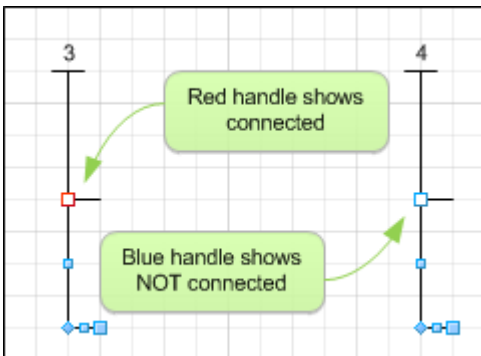
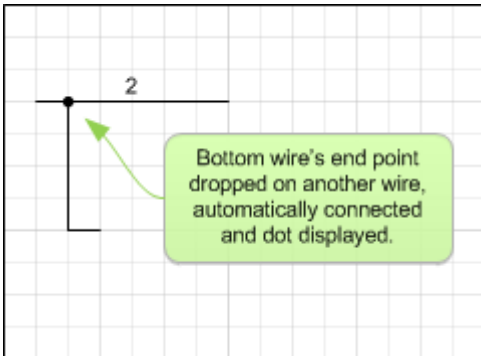
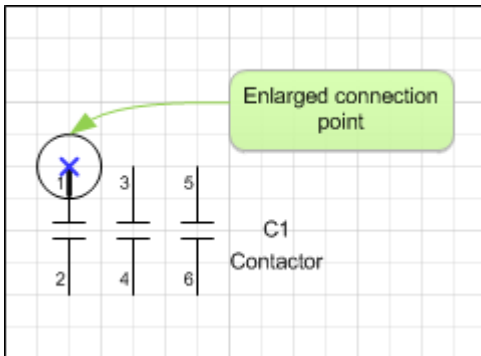


On the left of the Visio screen are a list of stencils, which contains a collection of shapes that you can drag and drop onto your drawings. The shapes on a stencil are called Master shapes or Masters. To select a stencil, click on its top bar. All stencils can be configured to display icons, icons and names or names only by clicking on the stencil icon (Right clicking in Visio 2010).

You can 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).

See Also: Placing and Duplicating Symbols, Connecting and Manipulating Wires

2.5 Connection Points



Most shapes in Electra have connections points to allow one shape to connect intelligently to another (e.g. connecting a 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 (In Visio 2010, they are dynamically hidden and shown). They also act as glue points to allow easy shape placements.

To add connection points:

- Click on a shape
- Click on the connection point tool
- Press CTRL key and click on anywhere you would like to add a connection point

To change the position of a connection point:

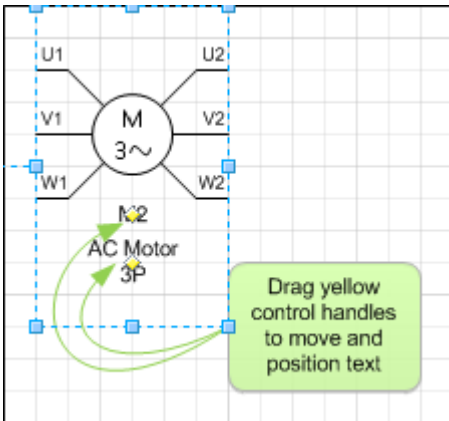
- Click on a shape
- Click on the connection point tool
- Click and drag a connection point from the shape to a new position

To delete a connection point:

- Click on a shape
- Click on the connection point tool
- Click on a connection point
- Press the DEL key

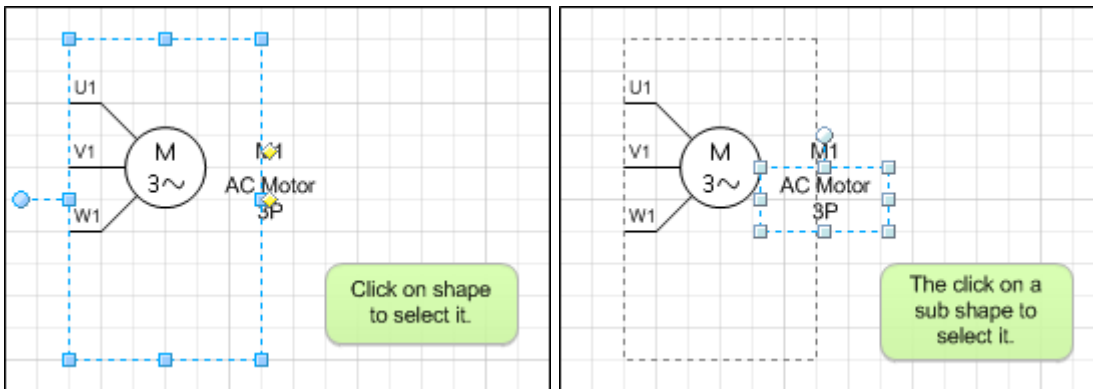
See Also: Automatic Wire Connection

2.6 Positioning Text



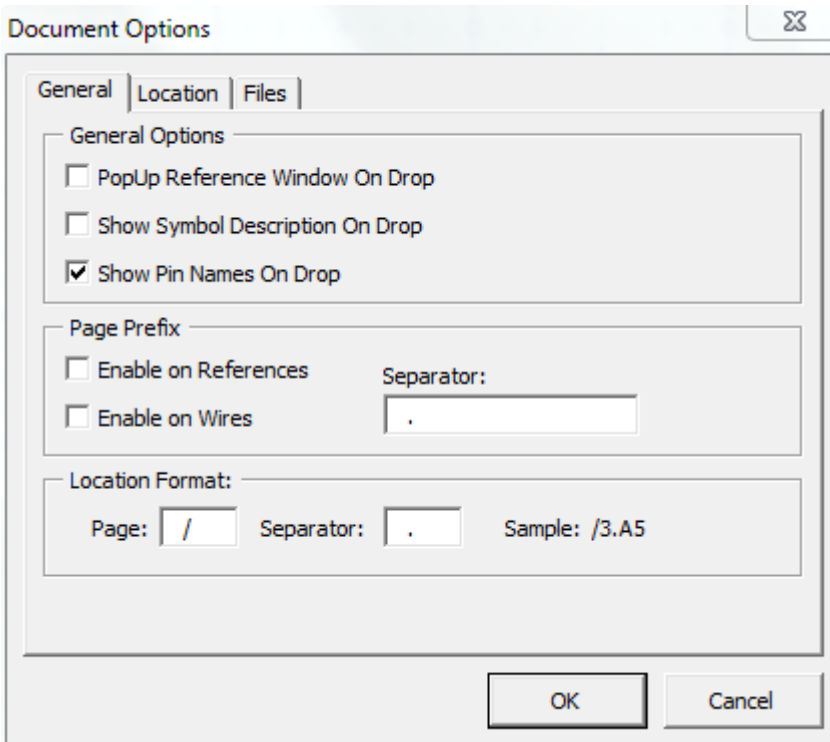
Symbols and wires in Electra have yellow control handles that let users easily re-position text. Simply drag on yellow control handles to move them to a new position.

2.7 Selecting Sub Shapes in a Group



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

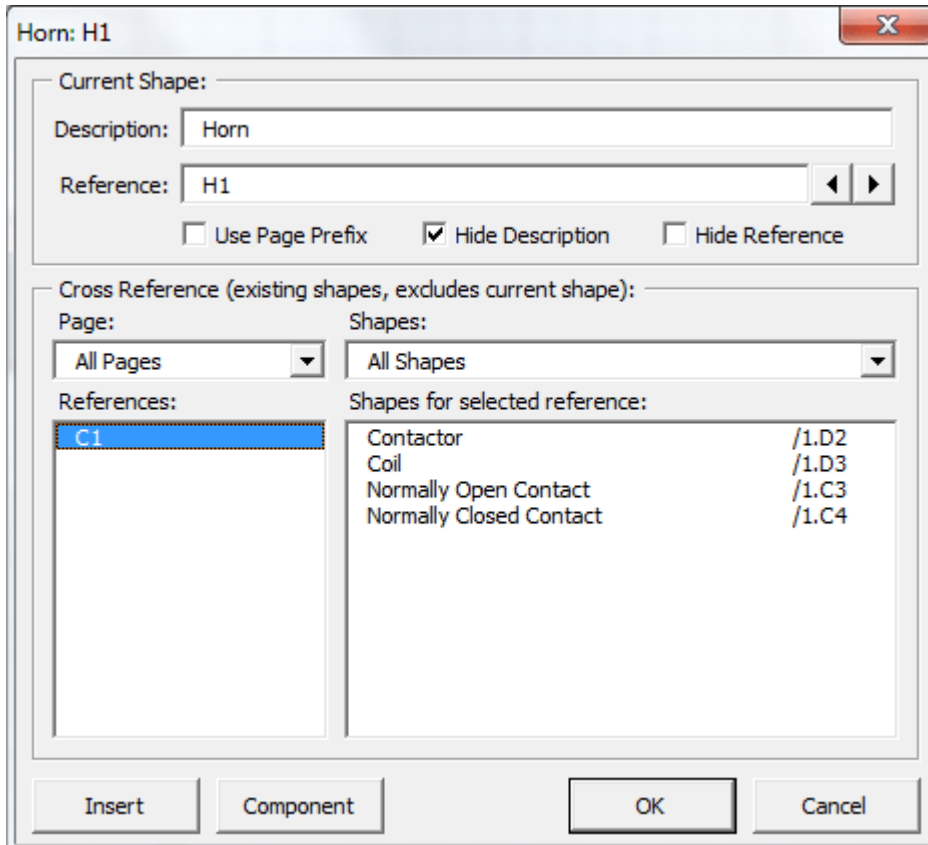
3. Document Options



Electra provides the user with drawing options that can be accessed by:

- Click menu Electra | Document Options (Visio 2010: Add-ins | Electra | Document Options)

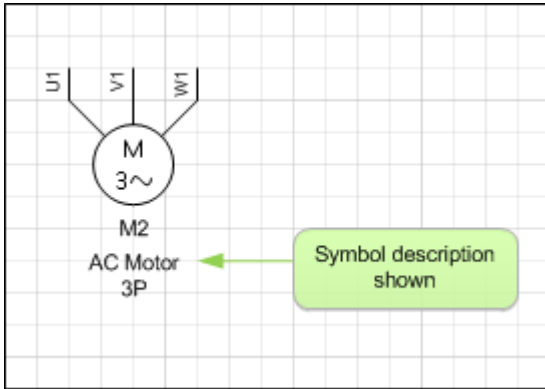
3.1 Popup Reference Window On Drop



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

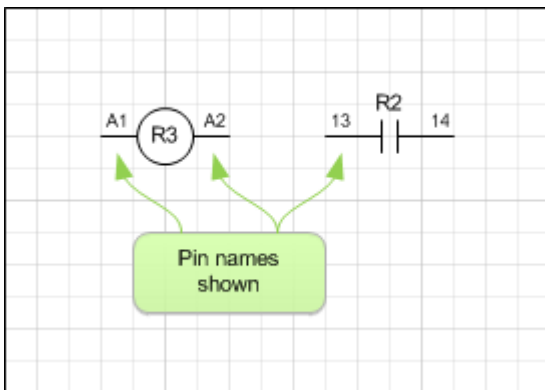
For faster schematic generation, uncheck the "Popup Reference window on drop" checkbox. The Reference window will still be accessible by right clicking on a symbol and selecting "Edit Reference" even when automatic popup has been disabled.

3.2 Show Symbol Description on Drop



User can select to show or hide symbol descriptions when dropping a symbol onto a drawing. To automatically show symbol descriptions, check the "Show Symbol Description on Drop" check box. Alternatively, right click on a symbol and select "Edit Reference" to show or hide symbol descriptions using the Reference window.

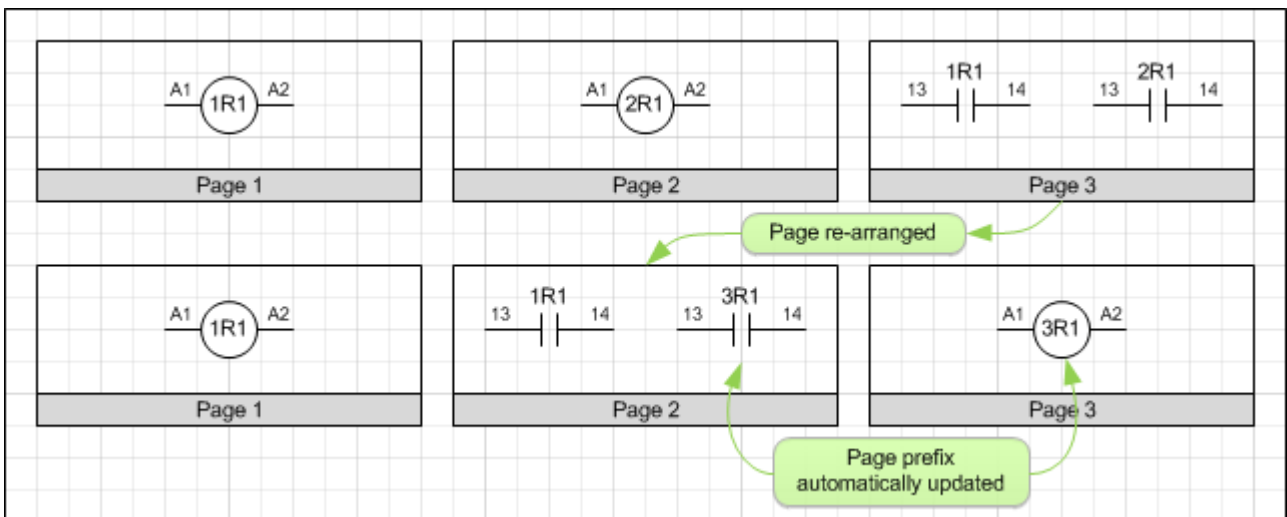
3.3 Show Pin Names on Drop



Users can select to show or hide Pin names when dropping a symbol onto a drawing. To automatically show pin names, check the "Show Pin Names on Drop" check box. Alternative, right click on a symbol and select "Set Pin Names" to show, hide or edit pin names.

See Also: *Automatic Pin Names, Assigning Pin Names*

3.4 How Page Prefix Works

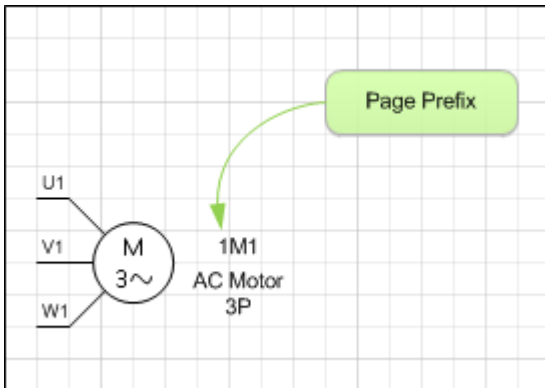


Page prefix in Electra is powerful and automatic. Once enabled, Page Prefix adds a page number in front of your symbol reference or wire name and automatically keeps track and updates this page number even when you re-order your pages, all automatically.

On the example above, there is a coil named 1R1 on page 1, another coil 2R1 on page 2 and 2 contacts, each of them belonging to the coil at page 1 and 2 respectively (top row of the image). When page 3 is reordered as page 2 (bottom row of image), the coil 2R1 previously at page 2 is

now automatically renamed to 3R1 on page 3. The contact 2R1 is also renamed to 3R1 since the coil it is belonging to is now renamed to 3R1 on page 3.

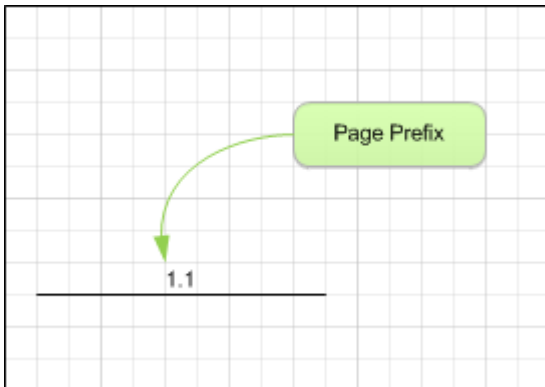
3.5 Enable Page Prefix on References



Users can select to prefix a page number on all references in a drawing by enabling the "Use Page Prefix on References" check box, and Electra will automatically insert and maintain all page prefixes. Alternatively, right click on a symbol and select "Edit Reference" to use Page Prefix.

See Also: [How Page Prefix Works](#)

3.6 Enable Page Prefix on Wires



Users can select to use page prefix for wires by enabling the "Use Page Prefix on Wires" check box and Electra will automatically insert and maintain all page prefixes. Alternatively, right click on a wire and select "Edit Wire" to use page prefix on a wire.

3.7 Page Prefix Separator for Wires

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

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

See Also: [How Page Prefix Works](#)

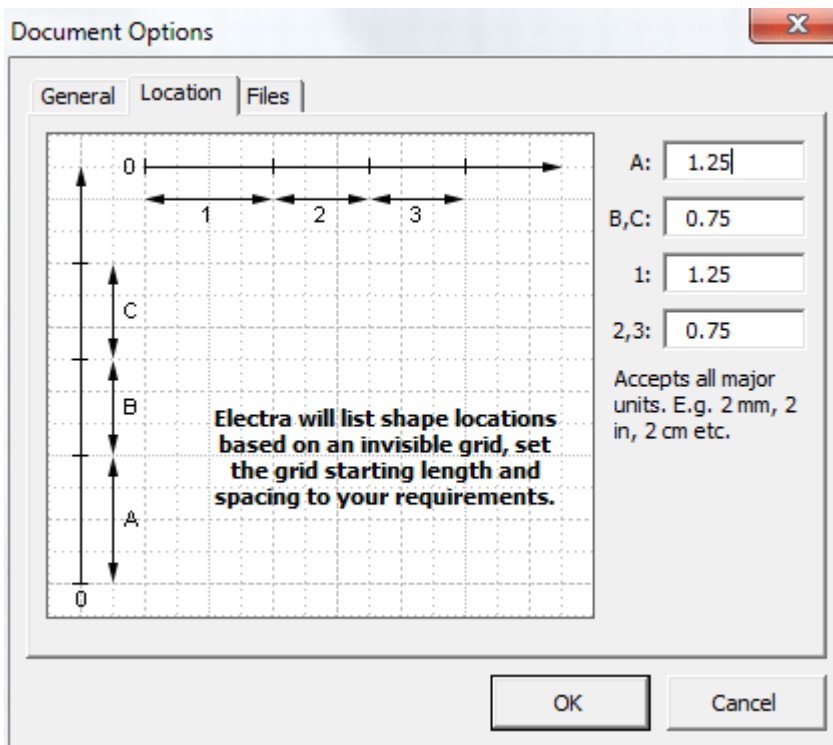
3.8 Location Format

Page	Separator	Examples
/	.	/3.E7, /4.F8
P	.	P3.E7, P4.F8
Pg	/	Pg3/E7, Pg4/F8

Users can change the format in which Electra display the location of your symbols and wires, as shown in the examples at left:

See Also: *The AutoLocation Shape*

3.9 Location Zone Options



Electra reports shape locations (e.g. /1.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 Also: *The AutoLocation Shape*

3.10 Saving Your Own Document Options

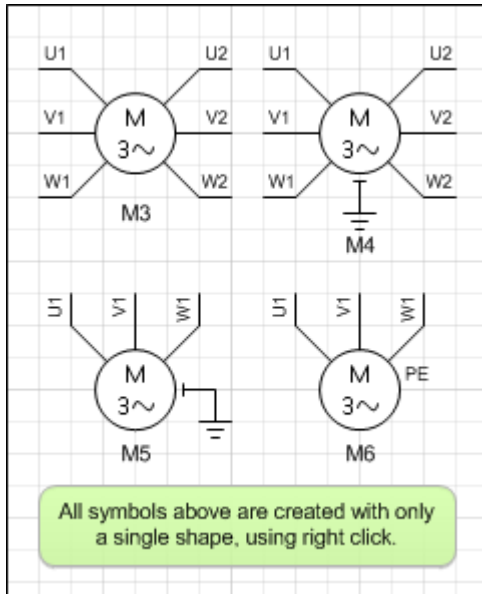
When you change the settings in Document Options, the settings will apply throughout the document (drawing file) and will be saved on the drawing file. If you wanted to set these settings as the default settings every time Electra is started, use the following steps:

- Make a backup copy of the file New Electra Drawing.vst (in C:\Program Files\Radica\Electra folder)
- Open Visio and Electra by double clicking on New Electra Drawing.vst.
- Select menu Electra | Document Options and apply your custom settings
- Do not modify the drawing, and select menu Save As
- Type in your custom template name or overwrite New Electra Template.vst
- Close Visio and restart

The settings are now saved into your own template. The next time you start Visio with your custom template, your settings will be used as the default.

4. Symbols and Components

4.1 One Intelligent Symbol



On some occasions, users may want to create drawings that display symbols horizontally. At other times, users may want to create vertical ones. Electra makes it easy to do both, with one single shape. Shapes in Electra are intelligent, and can be rotated vertically or horizontally with a right click. Rotated shapes automatically move text and descriptions to the correct position.

Certain shapes are even smarter. For example, the motor shape will allow users to place a 3 terminal or 6 terminal motor with a single right click. Drop a shape onto a drawing and right click to explore for more options.

4.2 Placing and Duplicating Symbols

To place a symbol:

- Drag and drop a symbol from a stencil

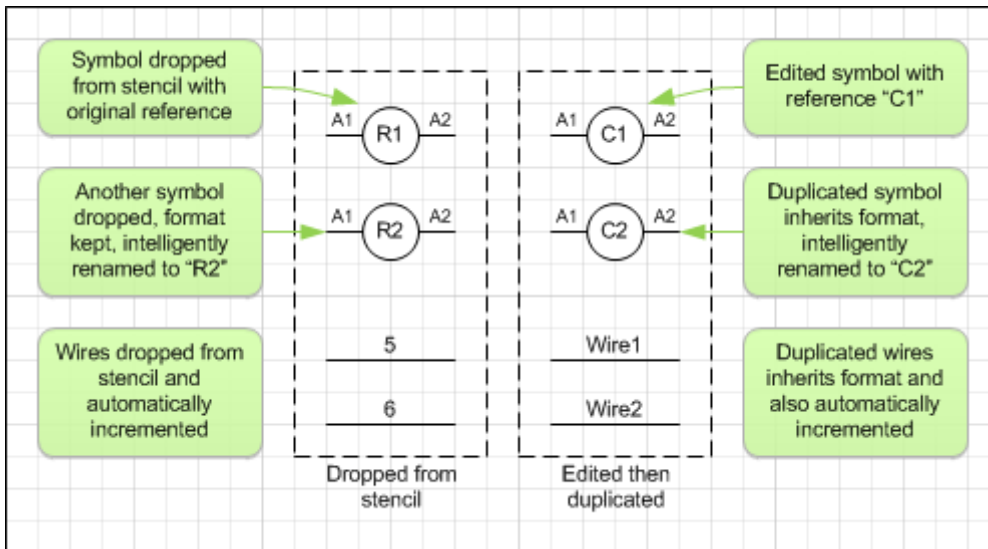
Symbols dropped from stencils inherits all formatting and data from its master shape in the stencil.

To duplicate a symbol (2 methods):

- Click on a symbol in a drawing, then click copy and paste
- Click on a symbol, press CTRL key, drag and drop, then release the CTRL key

Duplicated symbols inherits all formatting and data from the source symbol. Duplicating symbols are recommended when users wanted to preserve text or reference formatting.

4.3 Intelligent Automatic Numbering



Dropping a symbol by dragging from a stencil or by duplication and Electra will automatically and intelligently rename the new schematic symbol. If a symbol is dragged from a stencil, Electra will always rename the symbol by using its default reference. If a symbol is duplicated from an existing symbol, the new symbol will inherit all formatting and Electra will rename using the existing reference format.

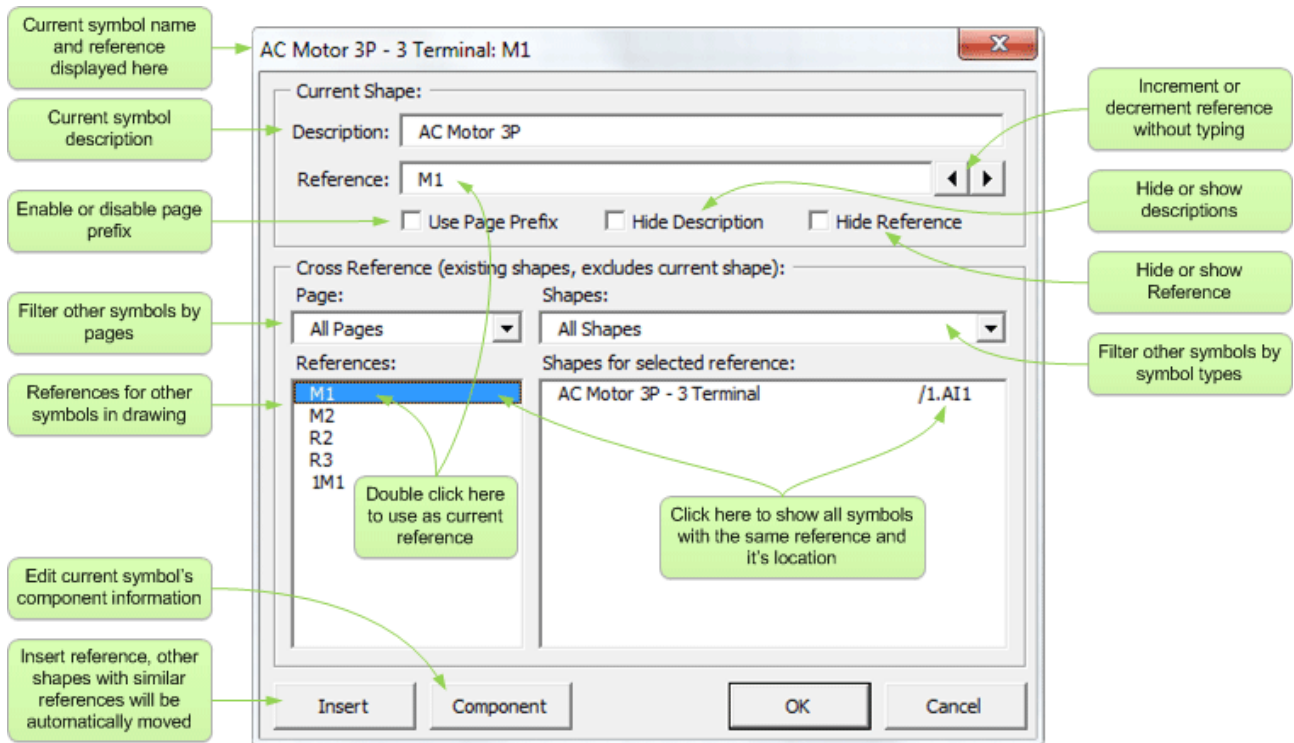
Create a new wire by dragging from a stencil or by duplication and Electra will automatically and intelligently rename the new wire. If a wire is dragged from a stencil, Electra will search through its internal sorted list and rename the new wire to the next sequential number. If a wire is duplicated from an existing wire, the new wire will inherit all formatting and Electra will rename using the existing wire name format.

4.4 Editing Symbol Reference

To edit the symbol reference (2 methods):

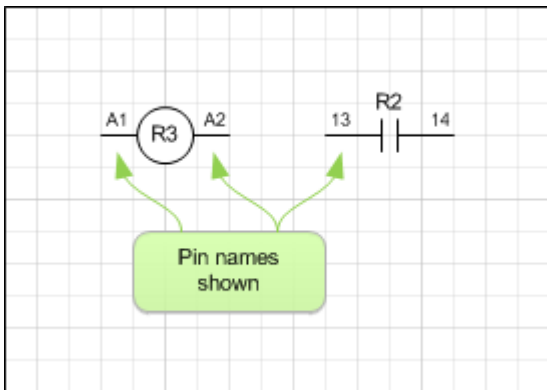
- Click on a symbol and type
- Double click on a symbol or Right click on a symbol and select "Edit Reference"

4.5 The Reference Window



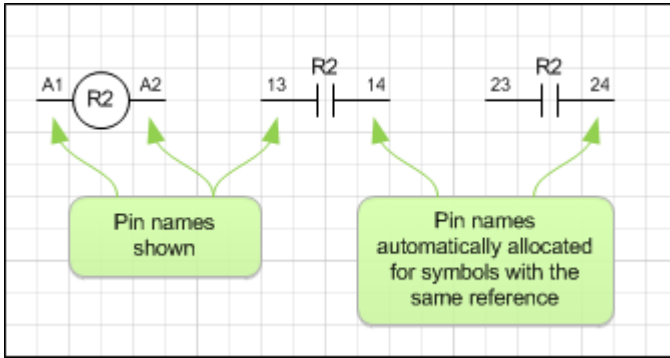
Included on the Reference window are real time cross reference information on shapes that you have already placed on your drawing. Click on a reference to display shapes and locations for that reference.

4.6 Displaying Pin Names



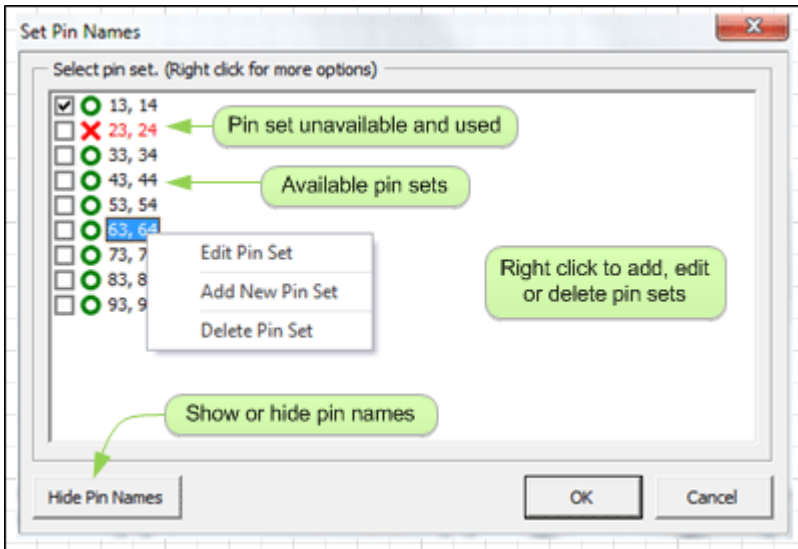
To display pin names by default when you drop a symbol, check the "Show Pin Names On Drop" checkbox under menu Electra | Document Options. Alternatively, right click on a symbol and select "Set Pin Names", then click on Hide or Show Pin names.

4.7 Automatic Pin Names



When symbols are dropped on a drawing, Electra automatically allocate the next available pin set to the symbol. When pin sets are not available or all used, Electra displays the pin names as red. Users can right click on a symbol and select "Set Pin Names" to manually allocate another pin set, show or hide pin names or add and edit pin sets.

4.8 Assigning Pin Names



To access the Pin Names window:

- Right click on a symbol, then select "Set Pin Names"

To assign another pin set:

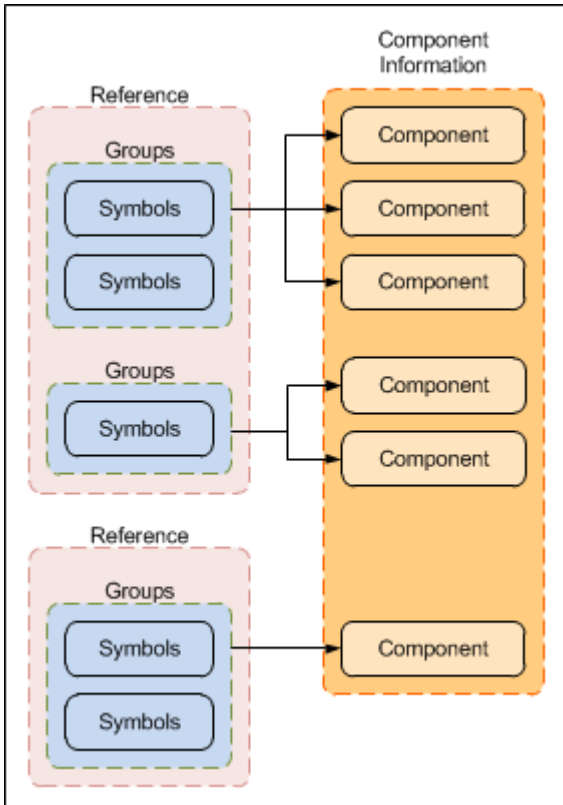
- Select a pin set then click OK.

To add, edit or delete pin set:

- Right click on any pin set and select your options

4.9 Understanding Symbols and Components

Symbols in Electra represents real world components, and the relationship can be any of the following:



Relationship	Example
One symbol to one component	A power supply symbol represents a real world power supply.
One symbol to many components	A relay symbol may represent a relay, a relay base and relay clips.
Many symbols to one component	Coil symbol, main contacts symbol and auxiliary contacts symbol all represent only a single contactor.
Many symbols to many components	Coil symbol and main contacts represents a contactor while multiple auxiliary contacts represents a single 4 pole auxiliary contact.
Note: All symbols have the same reference.	

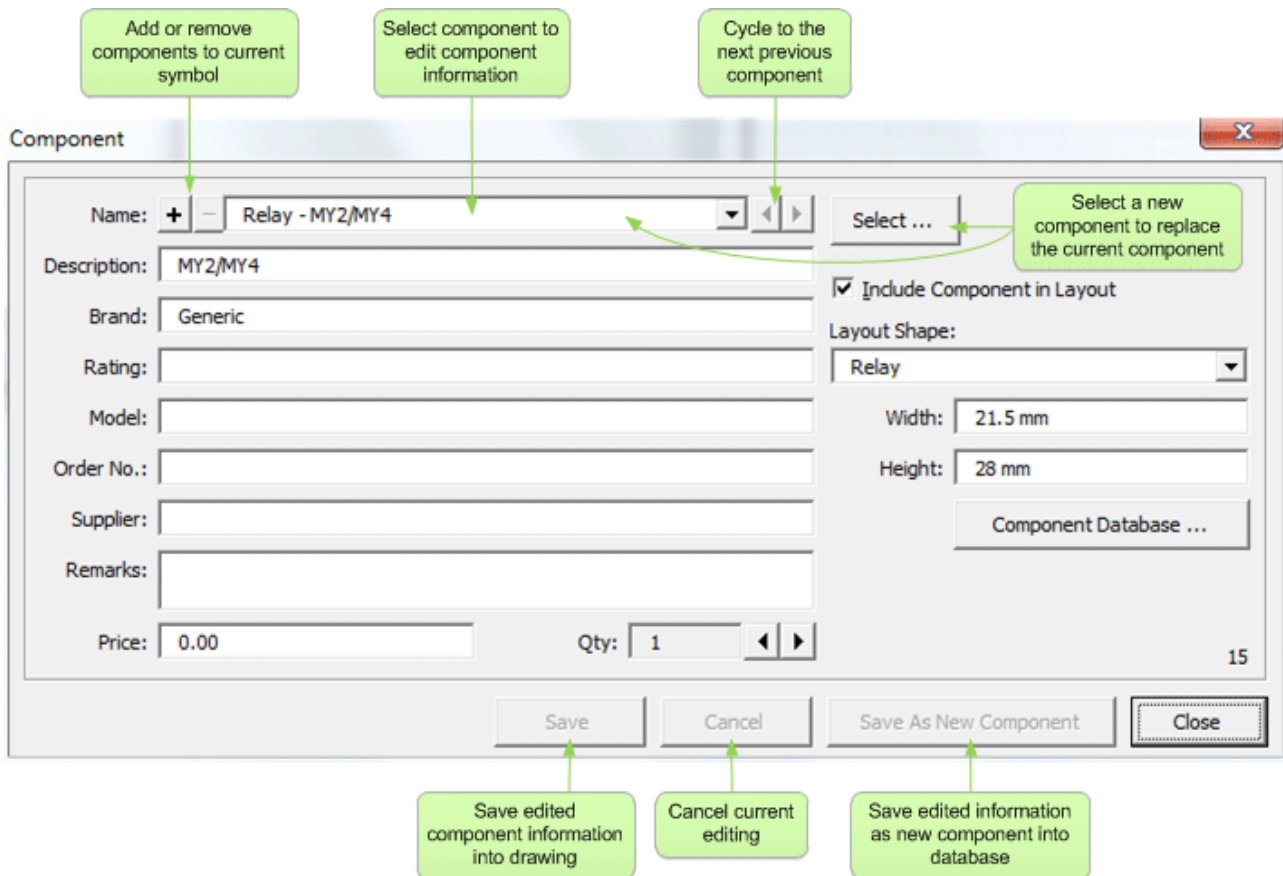
In Electra, all symbols with the same reference are initially organized into a single group, where they can be assigned to one or more components. Users can then divide the same symbols into further groupings, and each group can then be assigned to one or more components.

See Also: How Symbols are Matched to Components

4.10 Assigning Components

To assign a component to a symbol:

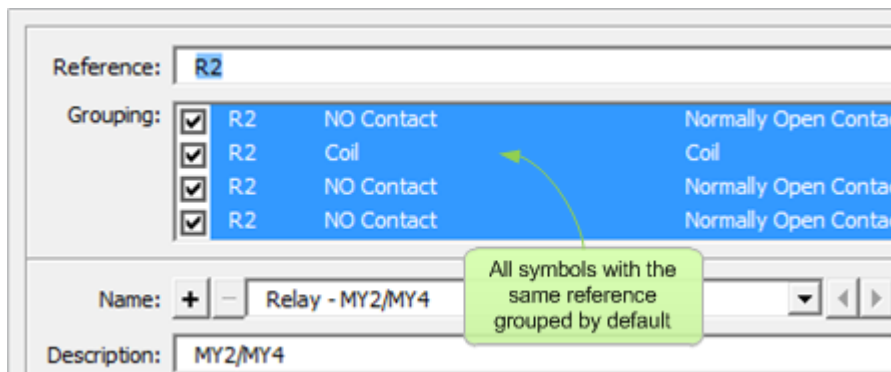
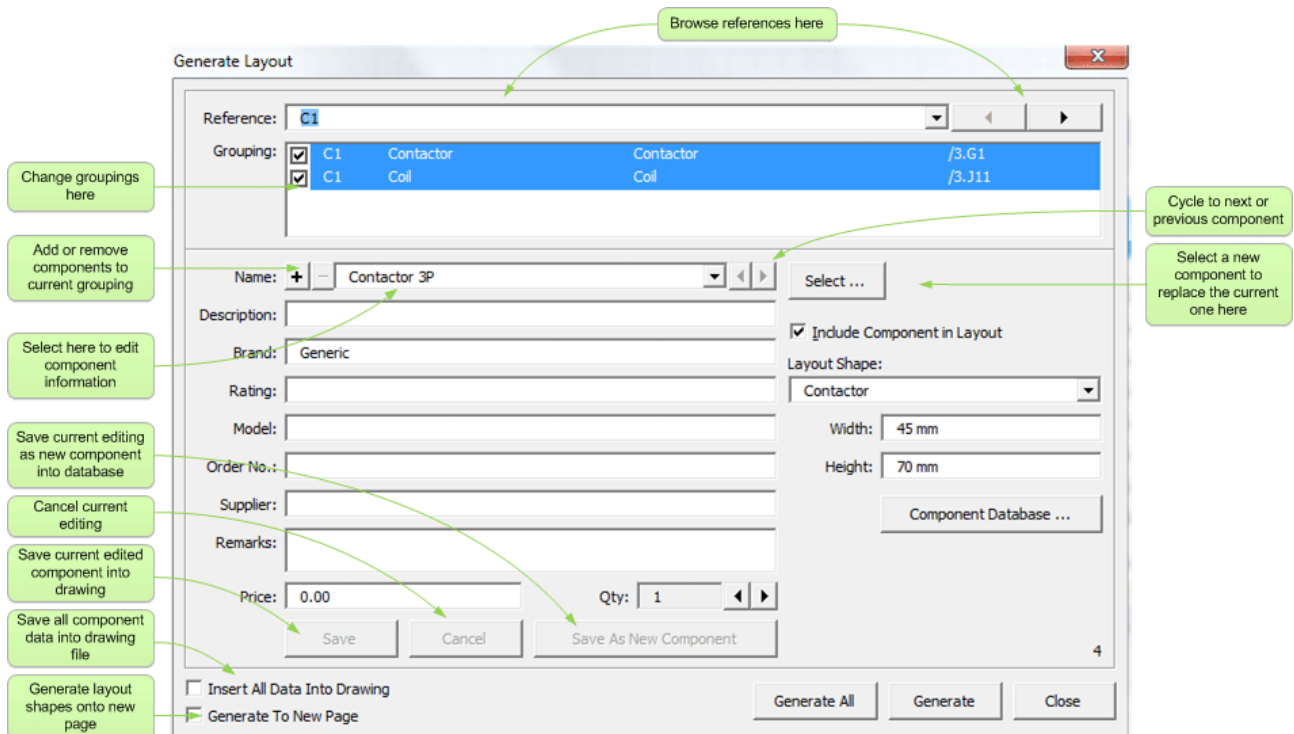
- Right click on a symbol and select "Edit Reference"
- On the Reference Window, click on "Component" button



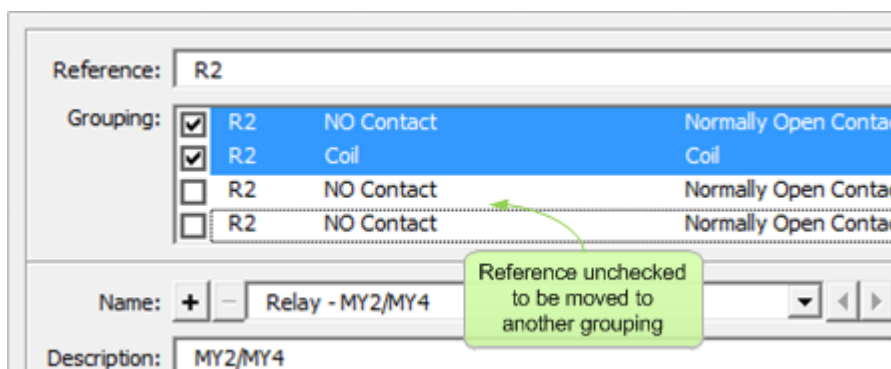
4.11 Managing Components

To assign and manage all symbols and components:

- Click on menu Electra | Manage Components



By default, all symbols with the same reference are grouped together. Users can then select to move some of the symbols to another group by unchecking the grouping box. Once unchecked, the symbols will automatically appear in another group ready to be assigned to another component.



To move back the symbols to its original group, simply select and uncheck the group.

Reference: **R2**

Grouping:

M1
M2
R2
R3
R2-Group2
1M1

New grouping automatically appears on reference drop down

Name: **+** **-** Relay - MY2/MY4

Description: MY2/MY4

Reference: **R2-Group2**

Grouping:

<input type="checkbox"/>	R2	NO Contact	Normally Open Contact
<input type="checkbox"/>	R2	Coil	Coil
<input checked="" type="checkbox"/>	R2	NO Contact	Normally Open Contact
<input checked="" type="checkbox"/>	R2	NO Contact	Normally Open Contact

Click on group to assign components to group

Name: **+** **-** Aux Contact

Description: 1NO/1NC Auxilliary Contact

Reference: **R2-Group2**

Grouping:

<input type="checkbox"/>	R2	NO Contact	Normally Open Contact
<input type="checkbox"/>	R2	Coil	Coil
<input checked="" type="checkbox"/>	R2	NO Contact	Normally Open Contact
<input checked="" type="checkbox"/>	R2	NO Contact	Normally Open Contact

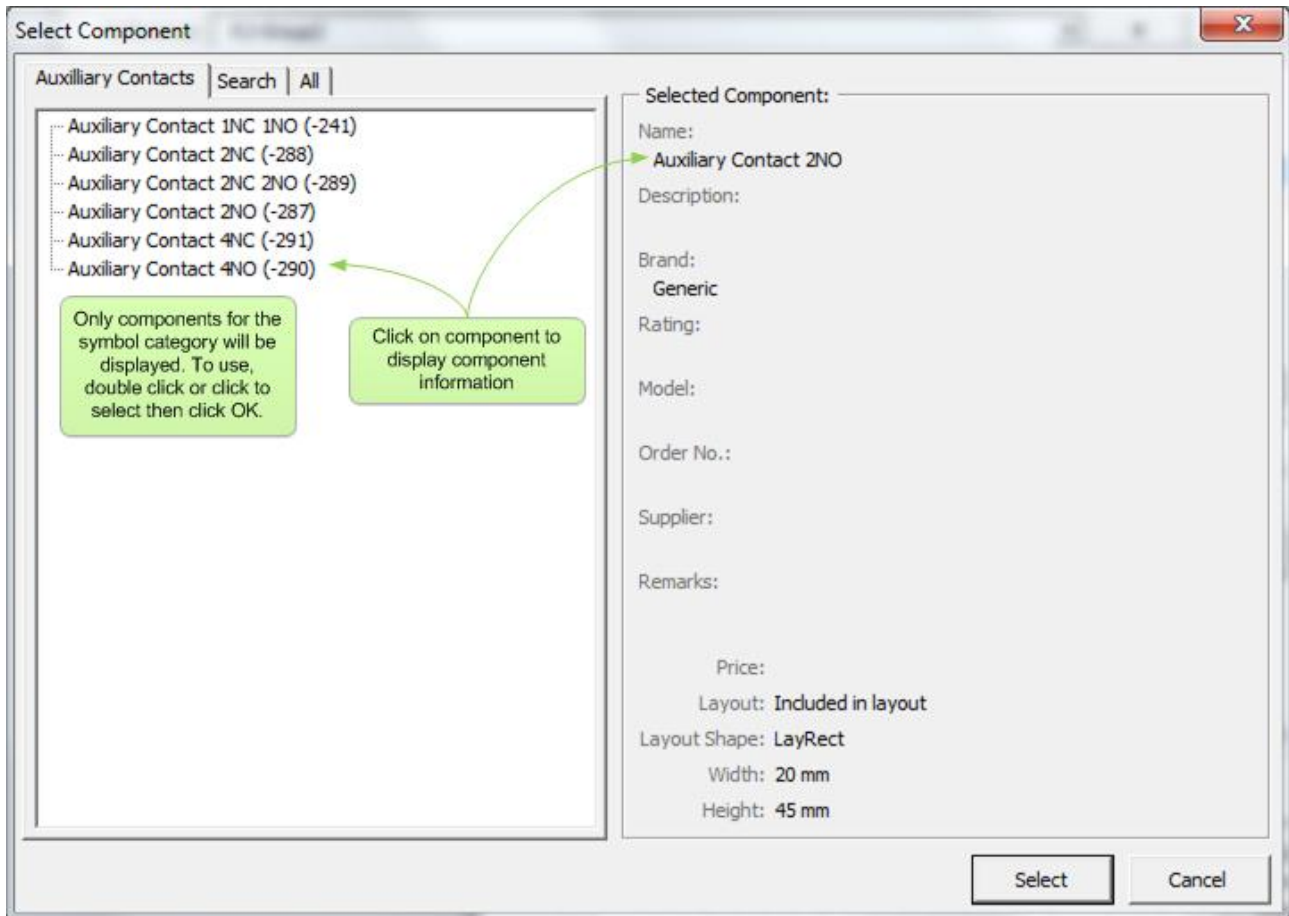
Uncheck to move back to original grouping

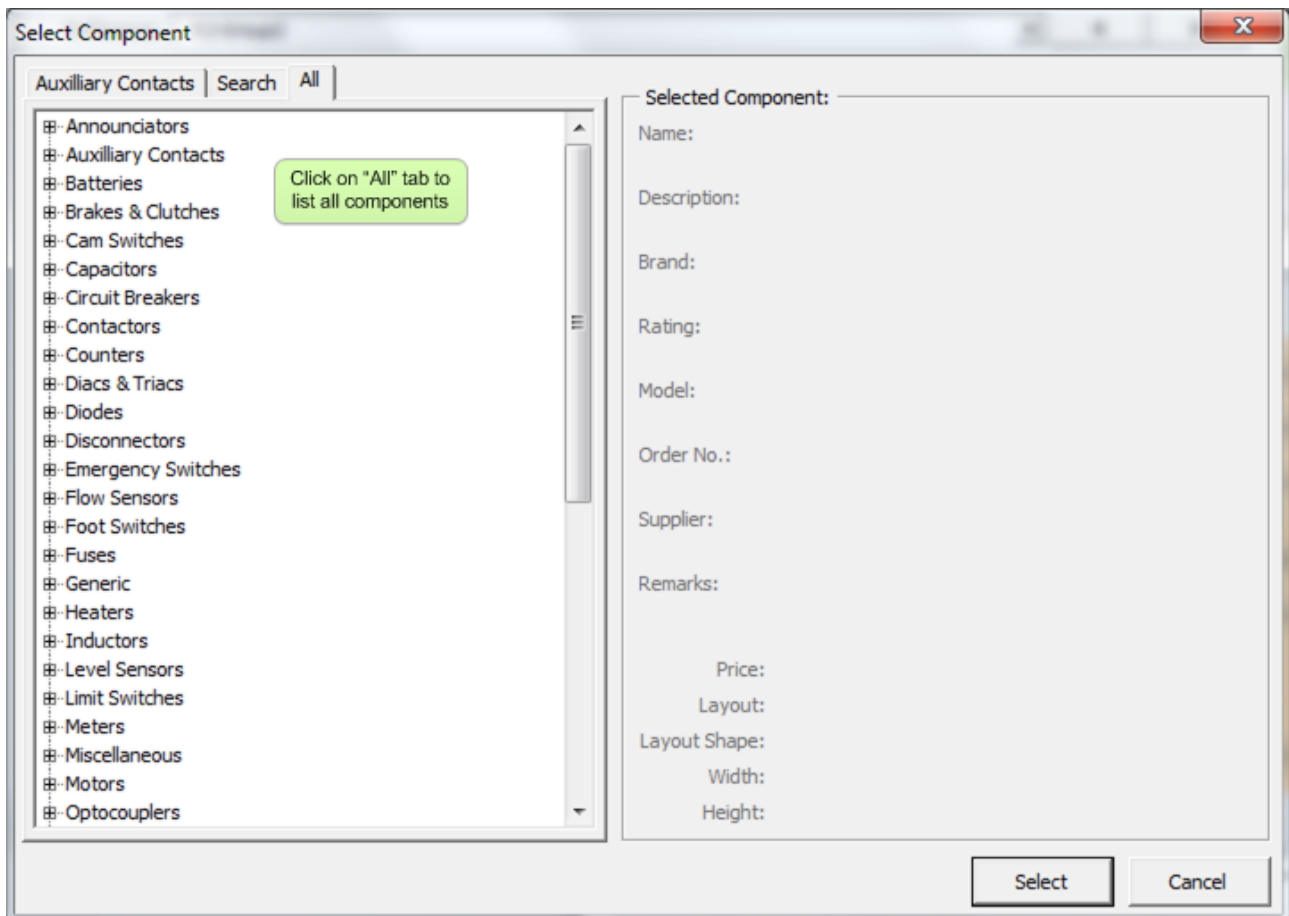
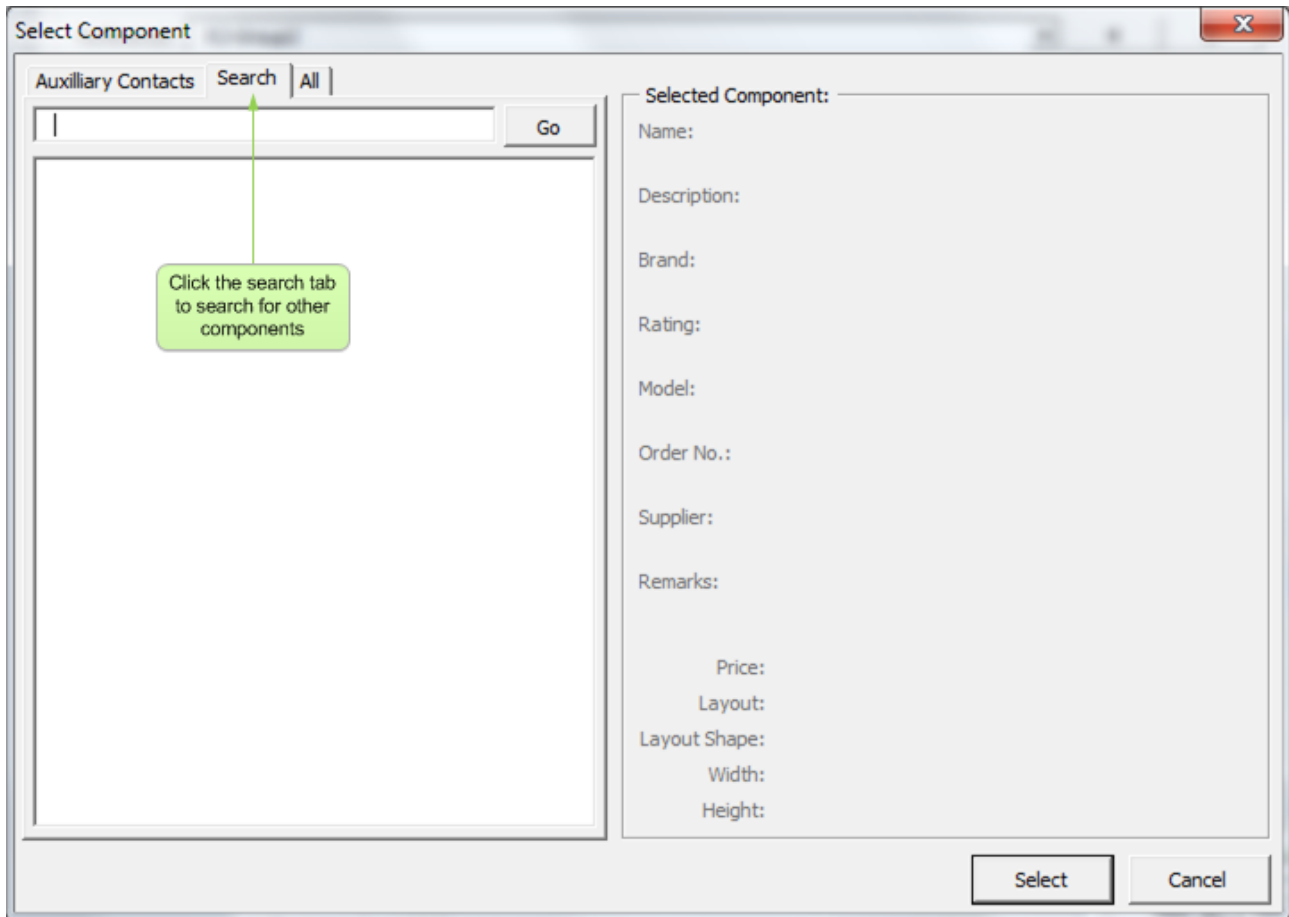
Name: **+** **-** Aux Contact

Description: 1NO/1NC Auxilliary Contact

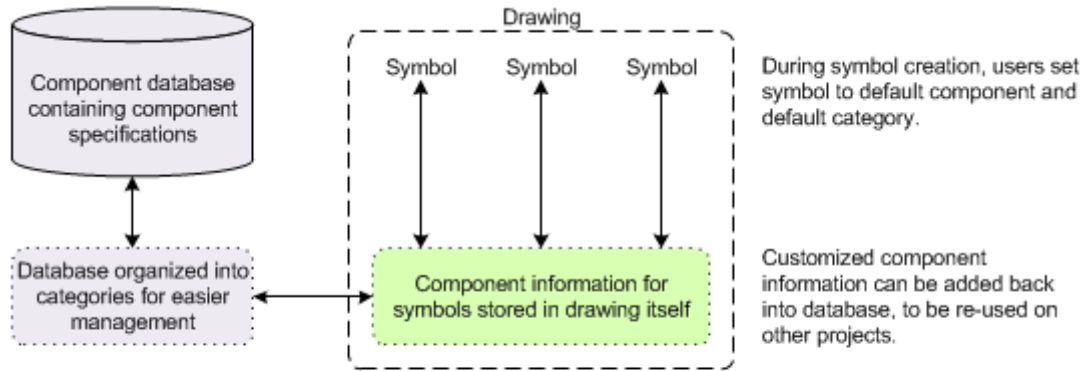
4.12 Selecting a Component

When users click the "Select" button on the Manage Components window, the Select Component window will be displayed:





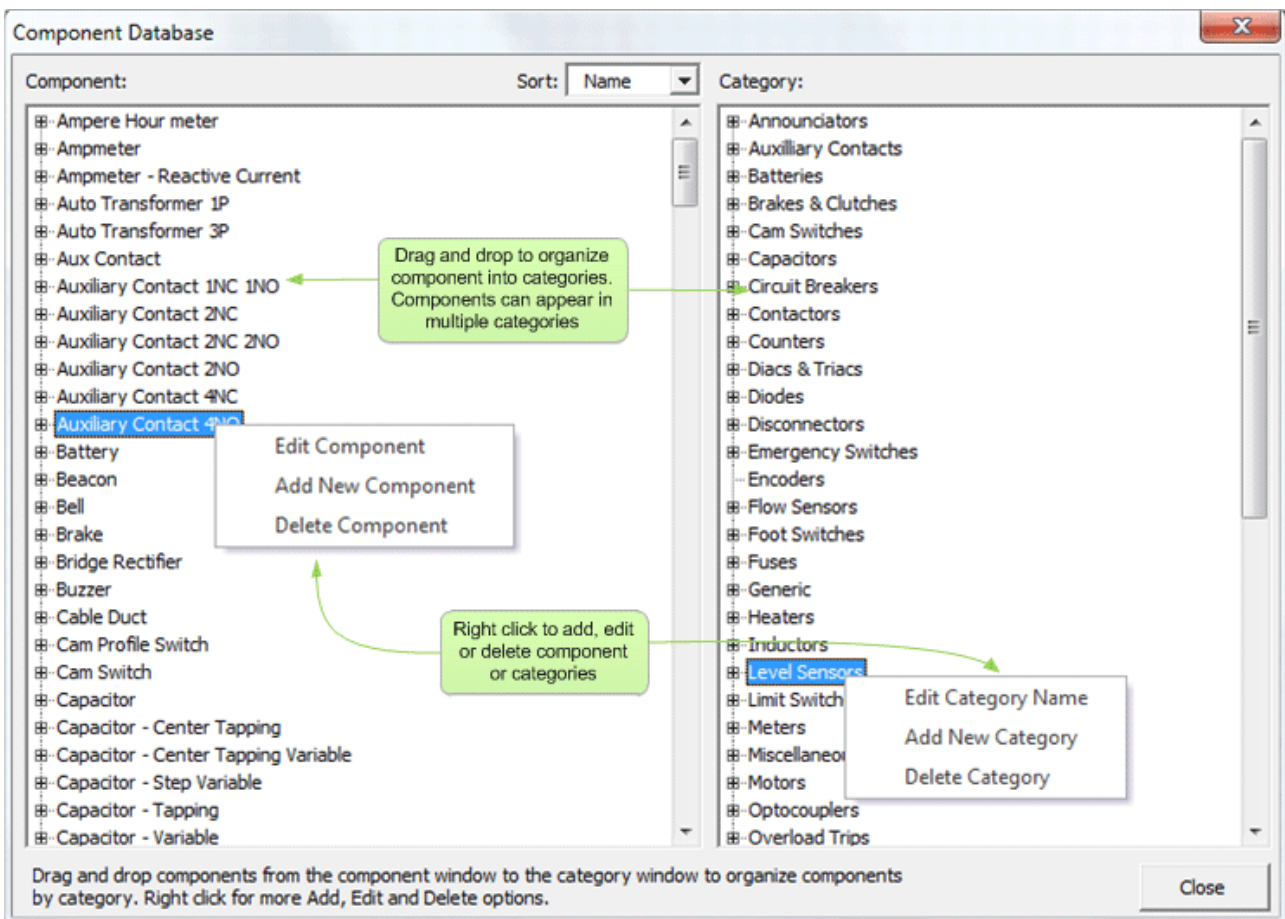
4.13 The Component Database



Users match and select components to be stored onto drawing. Only components from a single category will be presented to ease selection, although other categories and component searching are available.

To access the component database:

- Click on menu Electra | Component Database

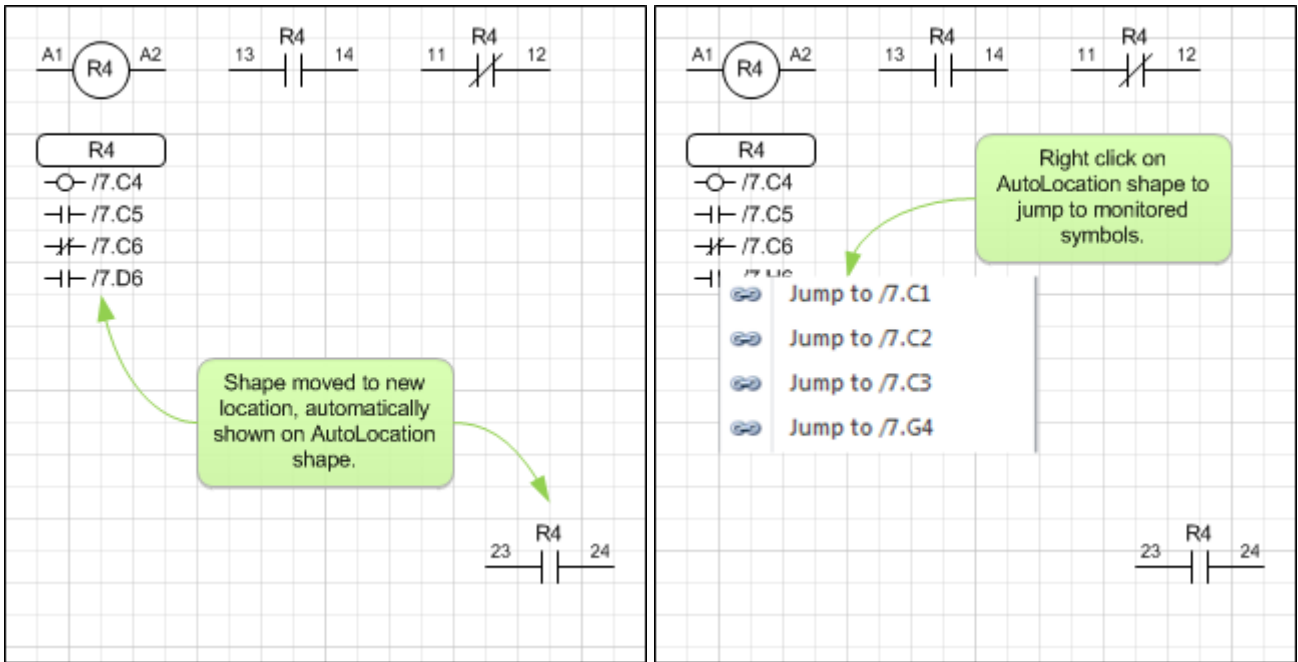


The Component database stores all component specifications so that they can be used and re-used on multiple projects and drawings. Once a component is selected, its specifications are then transferred and stored on the drawing itself for portability. Components stored on the database are divided into categories and a single component can appear in more than one category.

4.14 The AutoLocation Shape

The diagram illustrates the AutoLocation feature in a CAD software through four sequential panels:

- Panel 1 (Top Left):** Shows a drawing with three components: a circle labeled 'R4' with terminals 'A1' and 'A2', a resistor labeled 'R4' with terminals '13' and '14', and a switch labeled 'R4' with terminals '11' and '12'. A 'Ref' shape is placed on the drawing. A callout box states: "AutoLocation shape dropped on drawing, right click to locate." A toolbar icon for the 'Ref' shape is shown to the right.
- Panel 1 (Top Right):** Shows the 'Locate Reference' dialog box. The 'Reference:' field is empty. A list of components is displayed: C1, C2, M1, M2, M3, M4, M5, M6, MCB1, R1, R2, R3, R4, 1R1, 2R1. A callout box points to 'R4' in the list: "Double click or select a reference then click OK." The 'OK' and 'Cancel' buttons are at the bottom.
- Panel 2 (Bottom Left):** Shows the 'Ref' shape expanded to show a list of components: R4, -O- /7.C4, -| /7.C5, and -| /7.C6. A callout box points to the list: "Location of all shapes with selected reference displayed".
- Panel 2 (Bottom Right):** Shows the drawing with a new shape added: a resistor labeled 'R4' with terminals '23' and '24'. A callout box points to this new shape: "New shape added with the same reference". Another callout box points to the 'Ref' shape: "AutoLocation shape automatically adds and display location of new added shape".



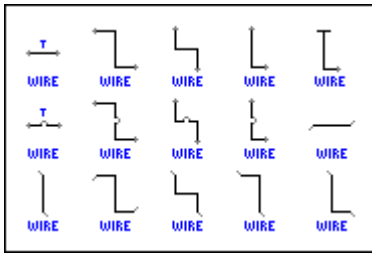
The AutoLocation displays the location of your selected reference automatically, as you add, modify or delete them, all without user interference. Locations will also update automatically when you move your symbols, even across multiple pages.

Right click on the AutoLocation shape to jump to monitored references. Users can place multiple AutoLocation shapes to monitor a single reference on multiple pages.

See Also: Location Format, Location Zone Options

5. Wires and Cables

5.1 A Variety of wires



Electra provides numerous types of wires for every conceivable type of connection. Each of them can be easily stretched and manipulated by dragging on their handles.

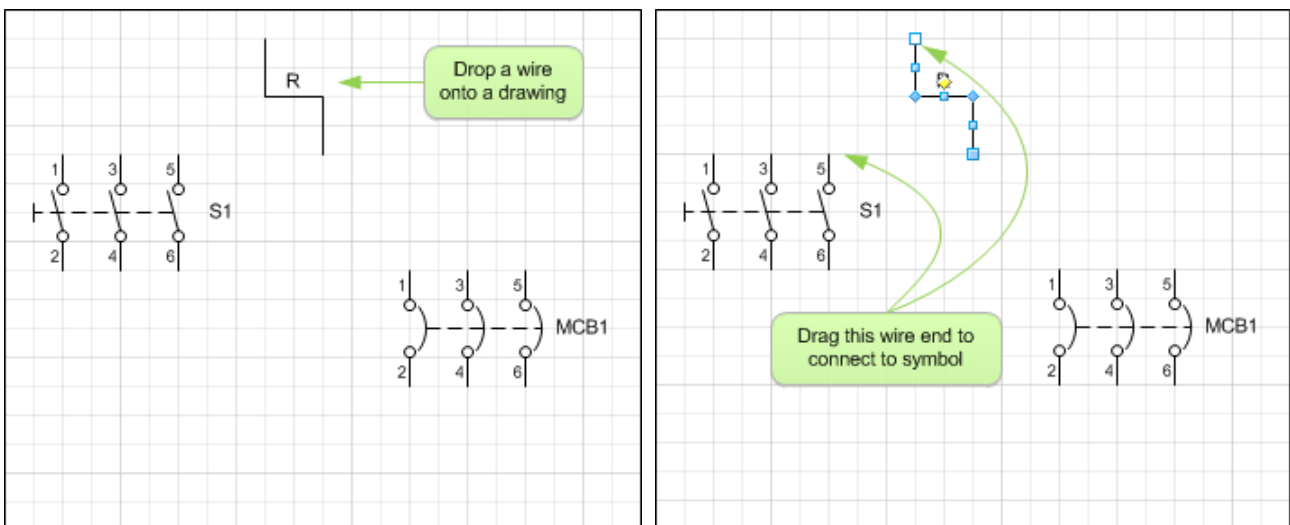
5.2 Difference Between A Line And A Wire

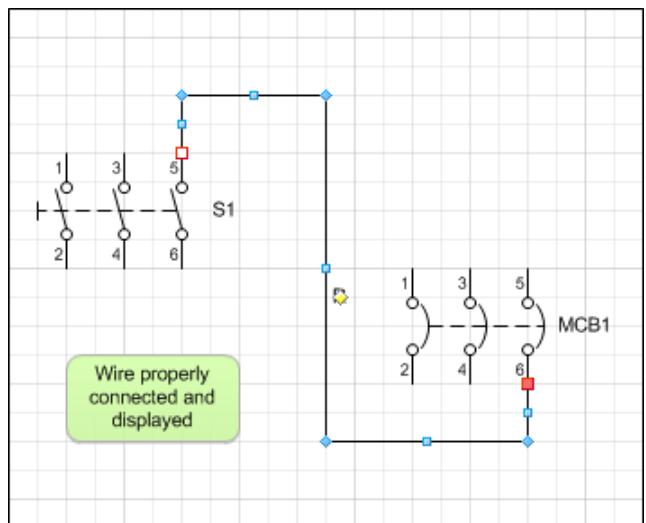
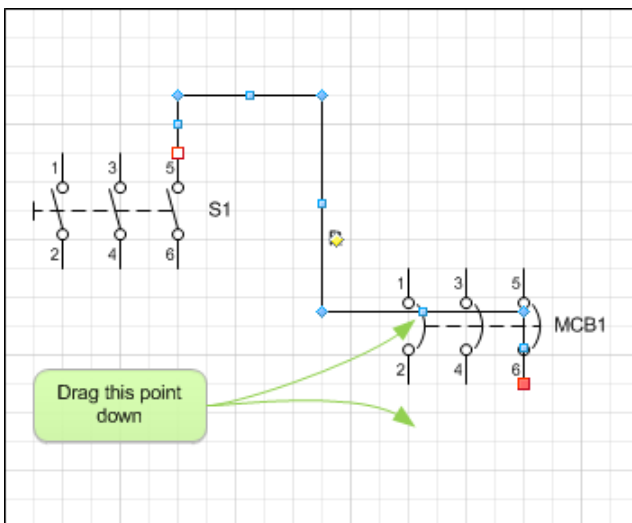
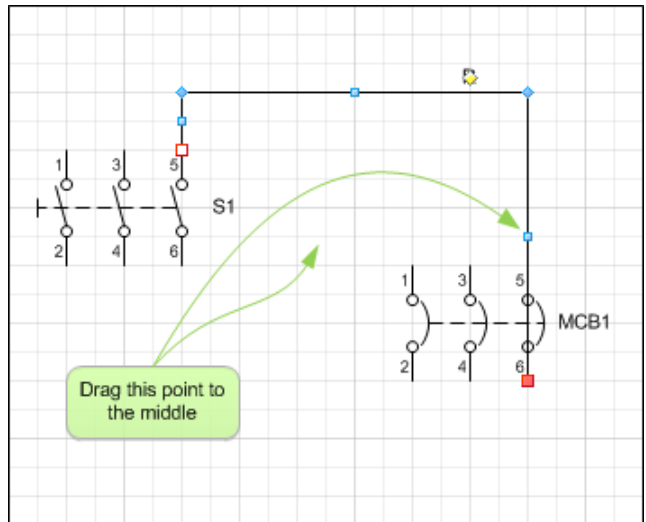
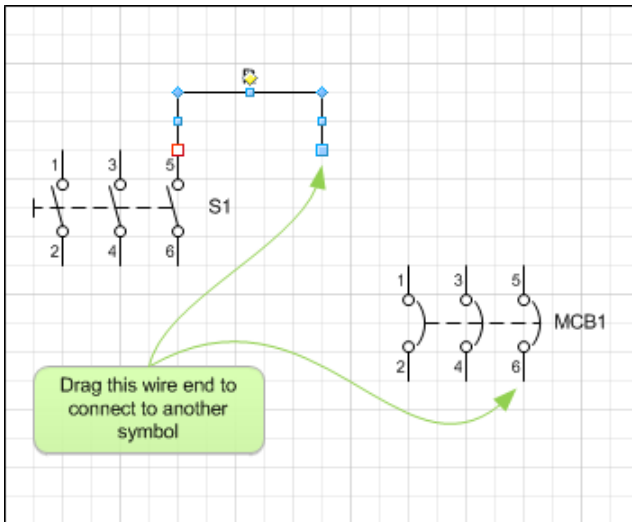
A normal line in Visio does not have data and intelligence as compared to a wire in Electra:

	Normal Visio Line	Electra wires
Automatic Wire Status Display (turns red)	No	Yes
Automatic Wire Name Detection	No	Yes
Automatic Wire Name Propagation	No	Yes
Automatic Wire Connection	No	Yes
Location Information	No	Yes
Right click for Dots	No	Yes
Detected by Terminals	No	Yes

5.3 Connecting and Manipulating Wires

Wires in Electra are extremely flexible and can be stretched and manipulated in a variety of ways, by dragging on their handles, as shown below:





5.4 Editing Wire Name

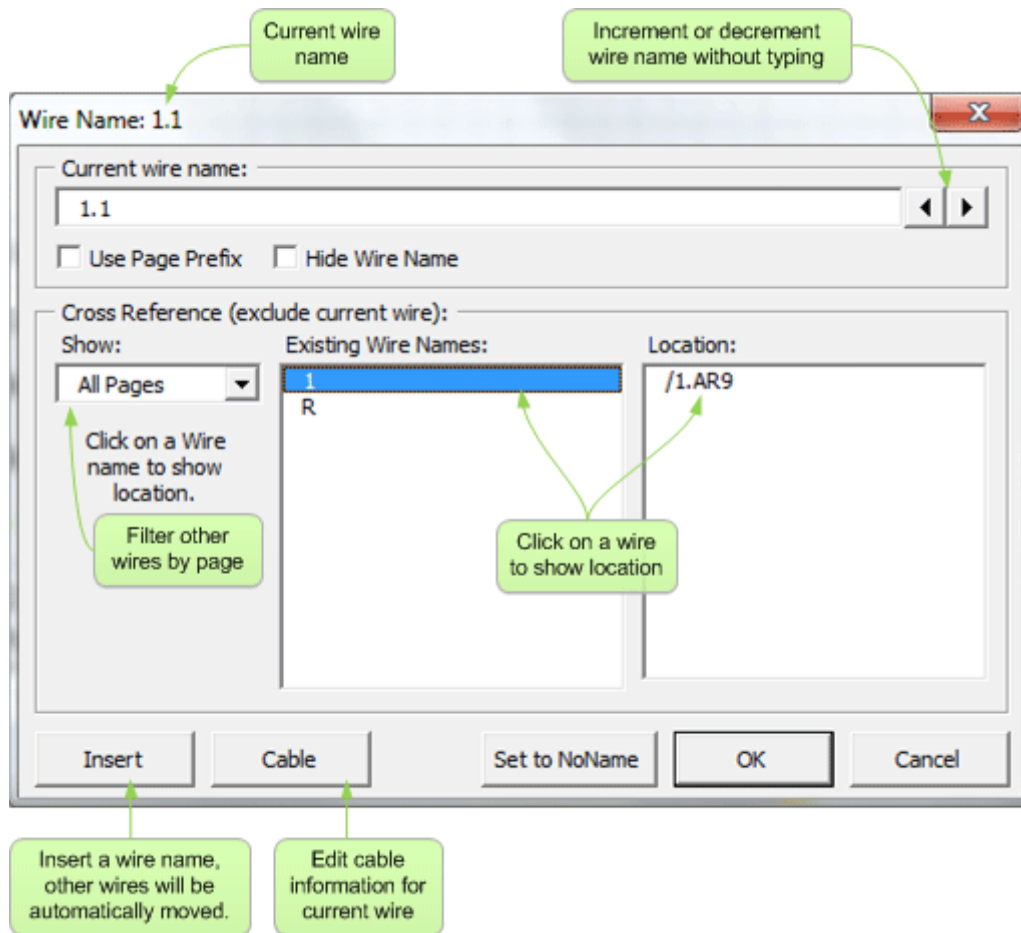
To edit a wire name (2 methods):

- Click on a wire and type
- Double click on a wire or Right click on a wire and select "Edit Wire"

To show or hide wire name:

- Right click on a wire and select Show or Hide Wire Name

5.5 The Wire Window

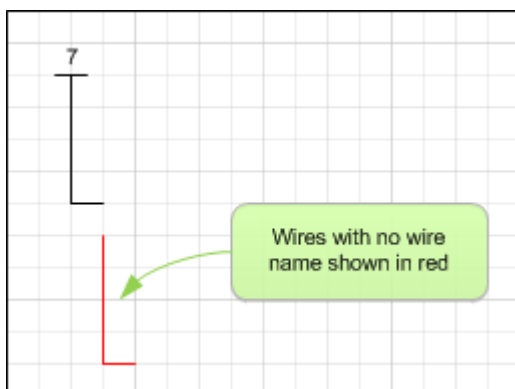


Included on the Wire window are real time cross reference information on wires that you have already placed on your drawing.

To display the location of a wire, click on a wire name. (e.g. /1.J1 - Page 1, location J1).

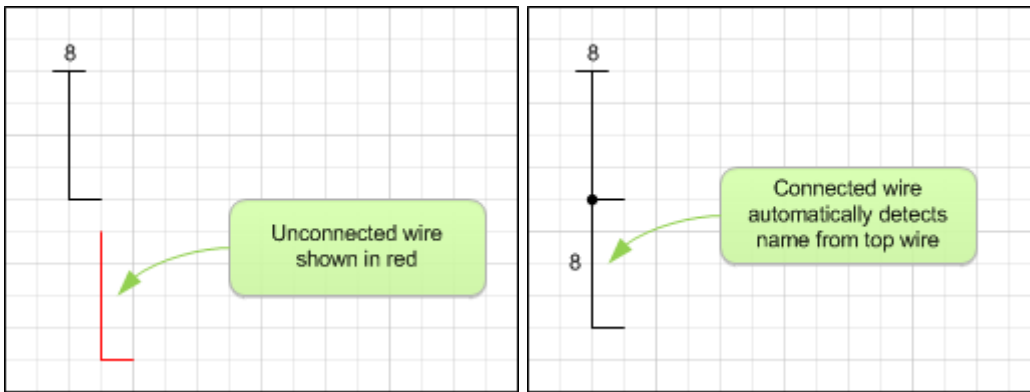
To set the current wire name to a wire name that already existed in your drawing, double click on a wire name in the Existing Wire Names list, and click OK.

5.6 Automatic Wire Status Display



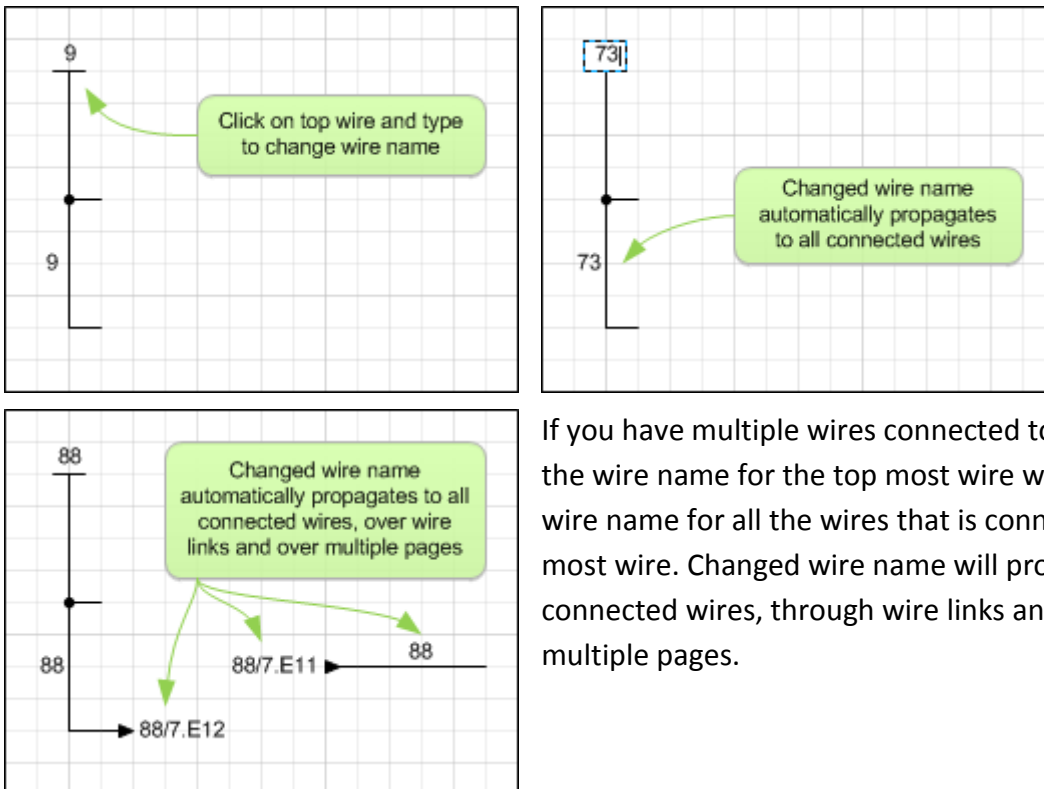
Wires in Electra will automatically be displayed in red when it does not have a wire name (orphaned). When a wire name is assigned to the wire, the wire will be displayed as black.

5.7 Automatic Wire Name Detection



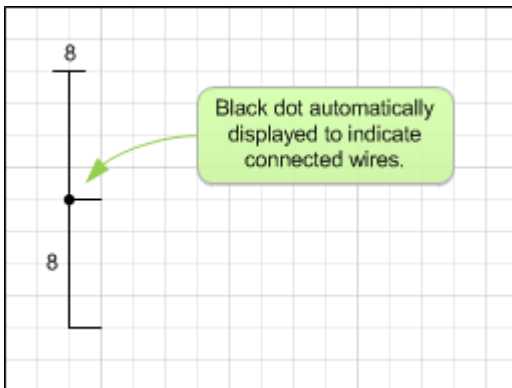
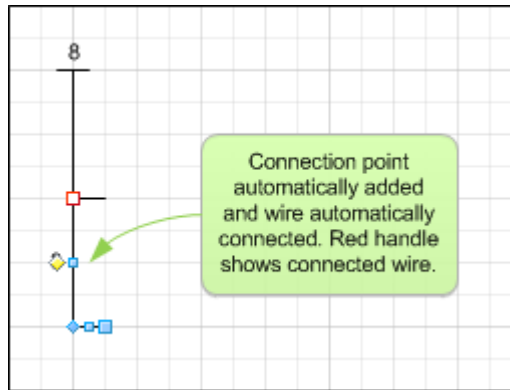
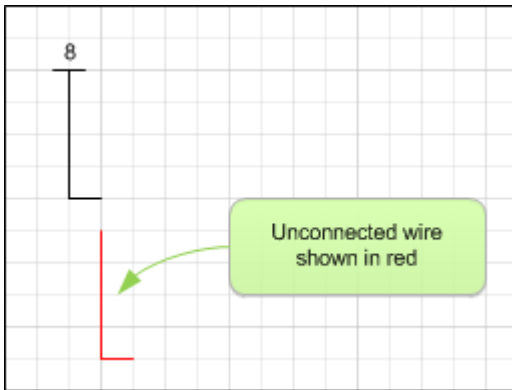
When a wire that has no wire name (red or orphaned) is dragged and moved to connect to another wire, Electra will automatically detect the new wire name and assign the new wire name to the orphaned wire.

5.8 Wire Name Propagation



If you have multiple wires connected together, changing the wire name for the top most wire will also change the wire name for all the wires that is connected to the top most wire. Changed wire name will propagate to all connected wires, through wire links and even over multiple pages.

5.9 Automatic Wire Connection

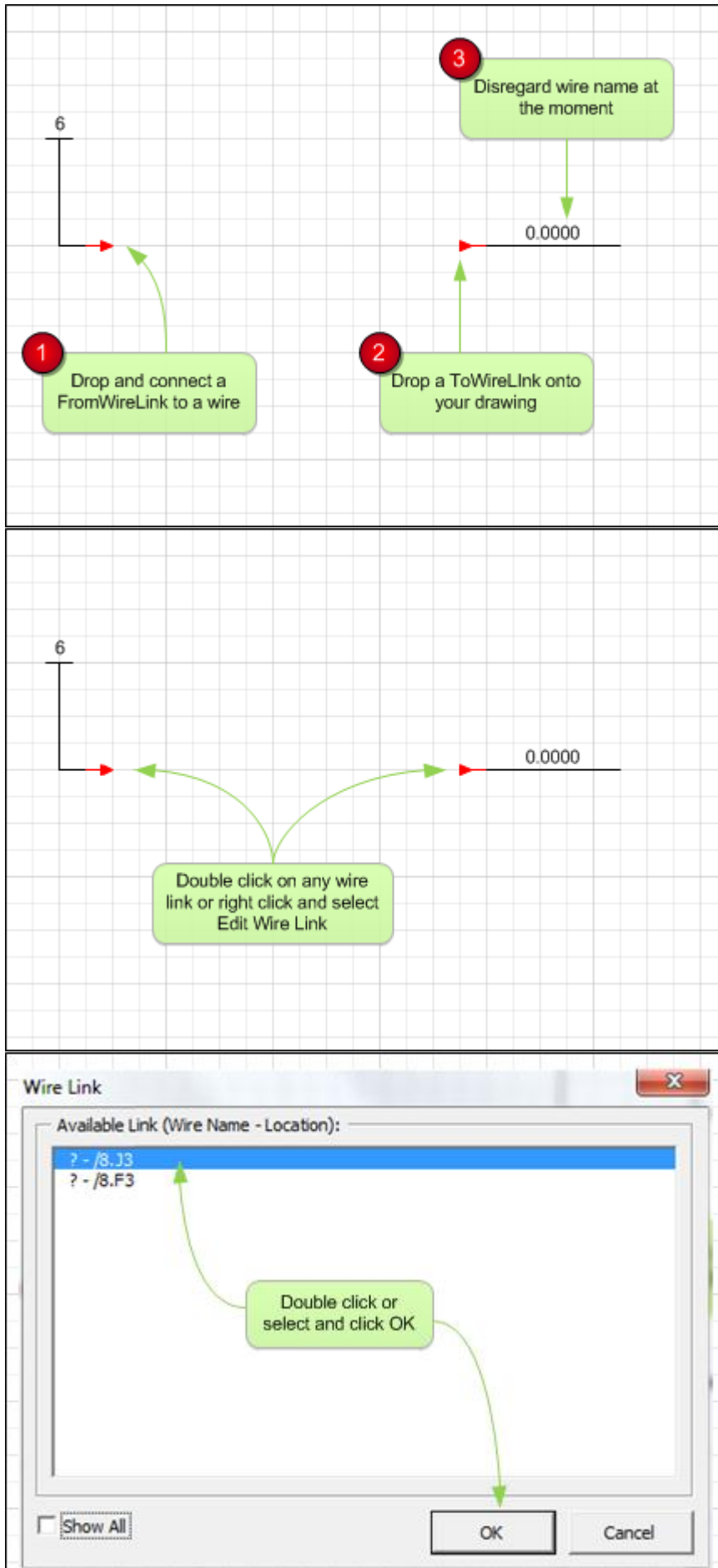


When a wire's end point is dropped on another wire, Electra automatically creates a connection point and connects the two wires together. When a connected wire is moved away, Electra automatically erases the connections points. Electra also automatically display a dot to denote a connection. Connection dots can be manually turned on/off by right clicking on the wire.

When clicking on a wire, the red handle at the wire end denotes a connection, while other colors shows that there is NO connection. A connection can only be made when a wire end is connected to a connection point.

See Also: Connection Points

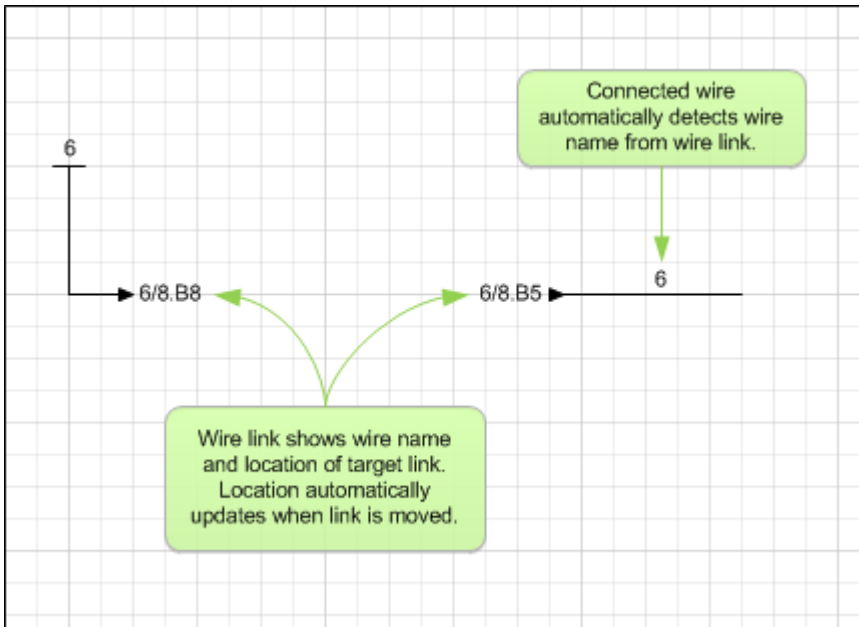
5.10 Wire links



Wire links are used to extend a wire to another location, on the same page or to another page, while still symbolically representing a single wire. Use FromWireLink (source) to extend a wire to another location and use ToWireLink (target) to continue a wire from another location.

Wire links can only be used in pairs, that is a source link (FromWireLink) that points to a target link (ToWireLink). Paired wire links always shows the location of the target paired link, and updates the location of the links automatically, even when moved.

Right click on wire links to edit or modify links, or to jump to a paired link.



5.11 Assigning Cables

To assign a cable to a wire:

- Right click on a wire and select "Edit Wire"
- On the Wire Window, click on "Cable" button

Cable

Click here to select existing cables on drawing or select "New Cable" to insert a new cable.

New Cable

Select...

If "New Cable" is selected, click here to select a cable to be inserted into drawing

Edit cable information here

Name: Default Cable

Brand: Generic

Model: Generic

Cores: 1

Core Name:

Arrange...

For multi-core cables, click here to assign and arrange cores

Core Names:

Comma separated, e.g: Red, Yellow, Blue

Area: 1.5

Order No.:

Supplier:

Remarks:

Price: 0

Save As New Cable

OK

Cancel

Cable

New Cable

Select...

Cores: 1, 1 Core, Generic, Generic, 2.5,
Cores: 1, 1 Core, Generic, Generic, 2.5,
Cores: 1, 1 Core, Generic, Generic, 2.5,
Cores: 3, Fan Motor 1, Generic, Generic, 2.5,
Cores: 3, Fan Motor 2, Generic, Generic, 2.5,
Cores: 3, Fan Motor 3, Generic, Generic, 2.5,
Cores: 3, Feeder Motor, Generic, Generic, 2.5,
Cores: 3, Main Conveyor, Generic, Generic, 3.5,

Core Names:

Comma separated, e.g: Red, Yellow, Blue

Area:

Order No.:

Supplier:

Remarks:

Price:

Arrange...

Save As New Cable

OK

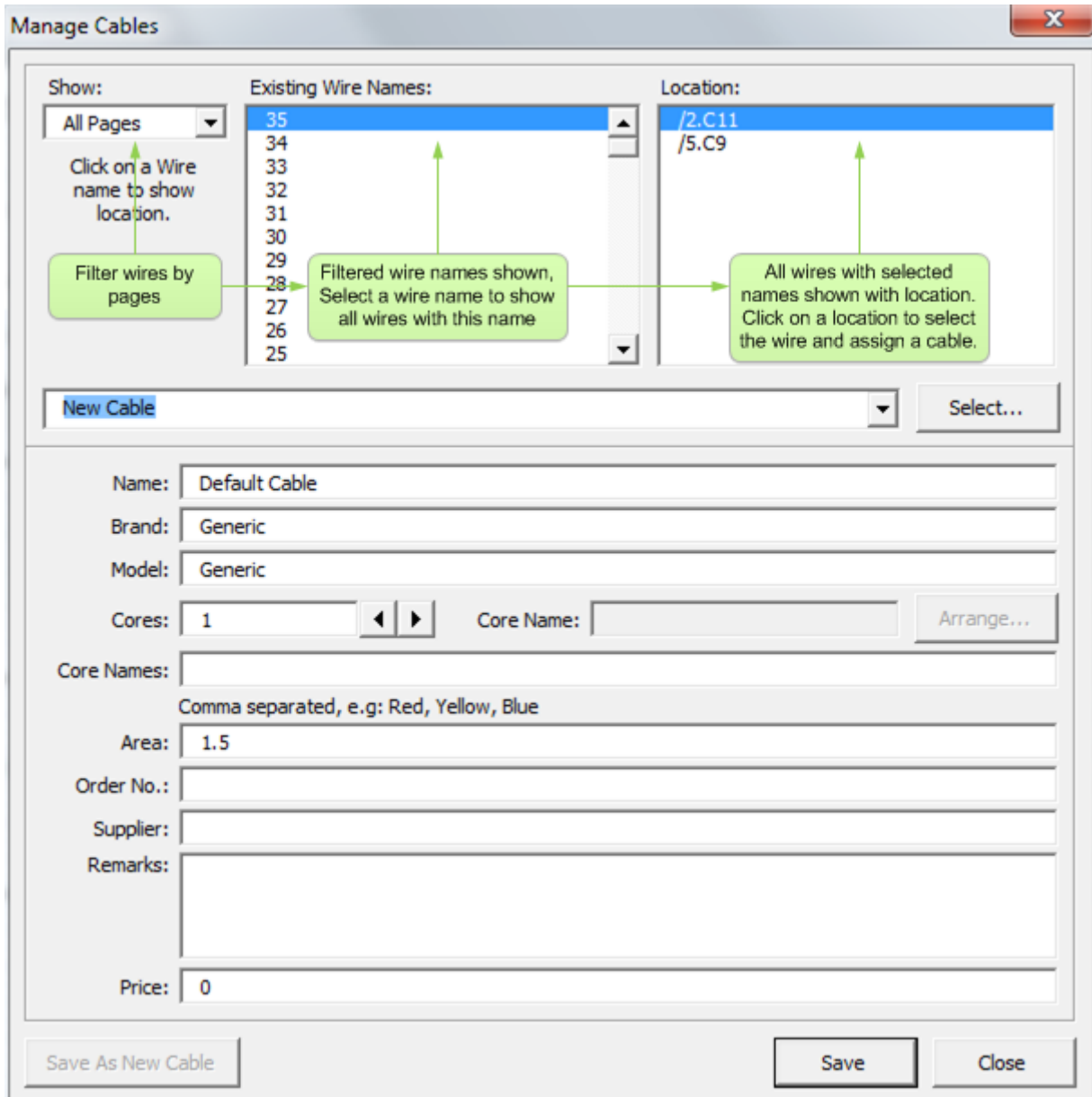
Cancel

Existing cables
already on drawing
shown here

5.12 Managing Cables

To assign and manage all cables:

- Click on menu Electra | Manage Cables



Manage Cables [X]

Show: **All Pages** [v]

Existing Wire Names:

35
34
33
32
31
30
29
28
27
26
25

Location:

/2.C11
/5.C9

Click on a Wire name to show location.

Click here to select existing cables on drawing or select "New cable" to insert a new cable.

If "New Cable" is selected, click here to select a new cable to insert into drawing.

New Cable [v] **Select...**

- Cores: 1, 1 Core, Generic, Generic, 2.5,
- Cores: 1, 1 Core, Generic, Generic, 2.5,
- Cores: 1, 1 Core, Generic, Generic, 2.5,
- Cores: 3, Fan Motor 1, Generic, Generic, 2.5,
- Cores: 3, Fan Motor 2, Generic, Generic, 2.5,
- Cores: 3, Fan Motor 3, Generic, Generic, 2.5,
- Cores: 3, Feeder Motor, Generic, Generic, 2.5,
- Cores: 3, Main Conveyor, Generic, Generic, 3.5,

Arrange...

Core Names: []
Comma separated, e.g: Red, Yellow, Blue

Area: 1.5

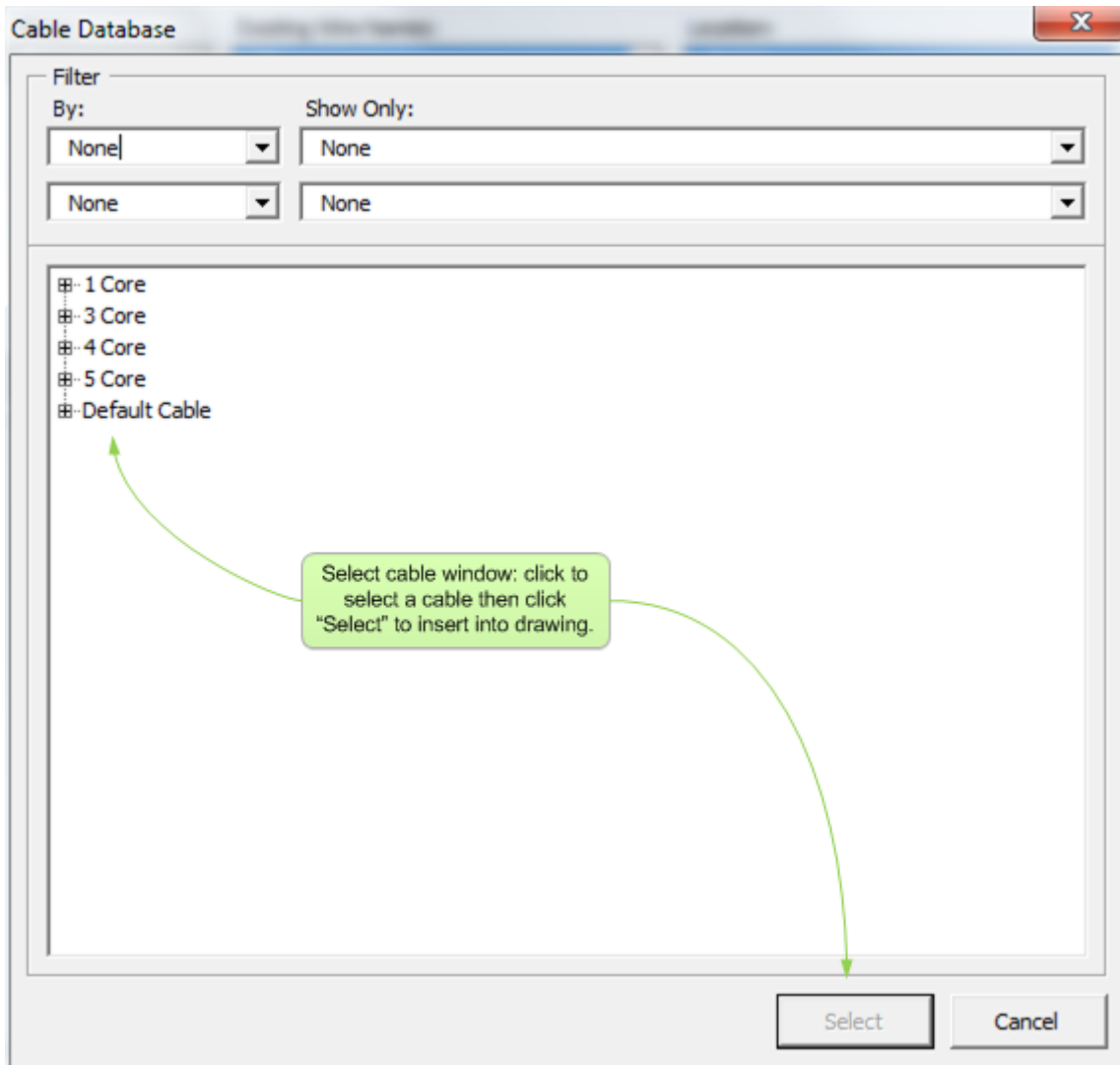
Order No.: []

Supplier: []

Remarks: []

Price: 0

Save As New Cable **Save** **Close**



Manage Cables X

Show:

All Pages ▾

Click on a Wire name to show location.

Existing Wire Names:

- V1
- U8
- U7
- U6
- U5
- U4
- U3
- U2
- U1
- T12
- T11

Location:

- /2.G6
- /2.G6

Cores: 3, Fan Motor 1, Generic, Generic, 2.5, Select...

Name:

Brand:

Model:

Cores: ◀ ▶ Core Name: Arrange...

Core Names:
Comma separated, e.g: Red, Yellow, Blue

Area:

Order No.:

Supplier:

Remarks:

Price:

Save As New Cable
Save
Close

After inserting cable into drawing, click "Arrange" to assign the current wire to another core, if using multi core cable.

Arrange Cable Cores

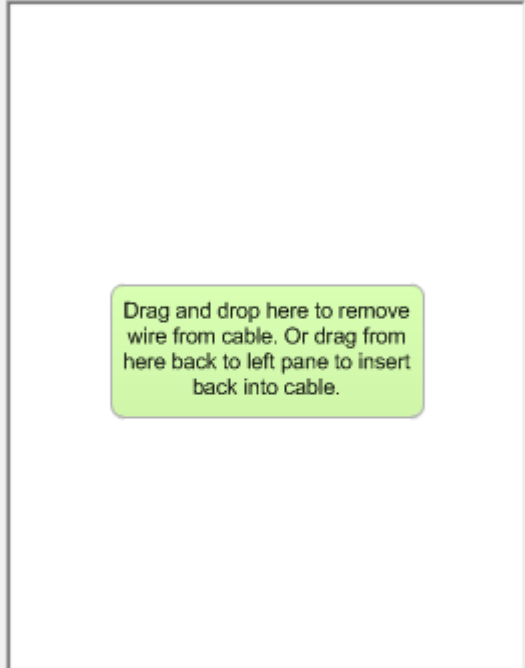


Core Name: Wire Name - Location

- Red: U1 - /2.G6
- Yellow: V1 - /2.G6
- Blue: W1 - /2.F6

Drag and drop to re-arrange wire and core names

Remove:



Drag and drop here to remove wire from cable. Or drag from here back to left pane to insert back into cable.

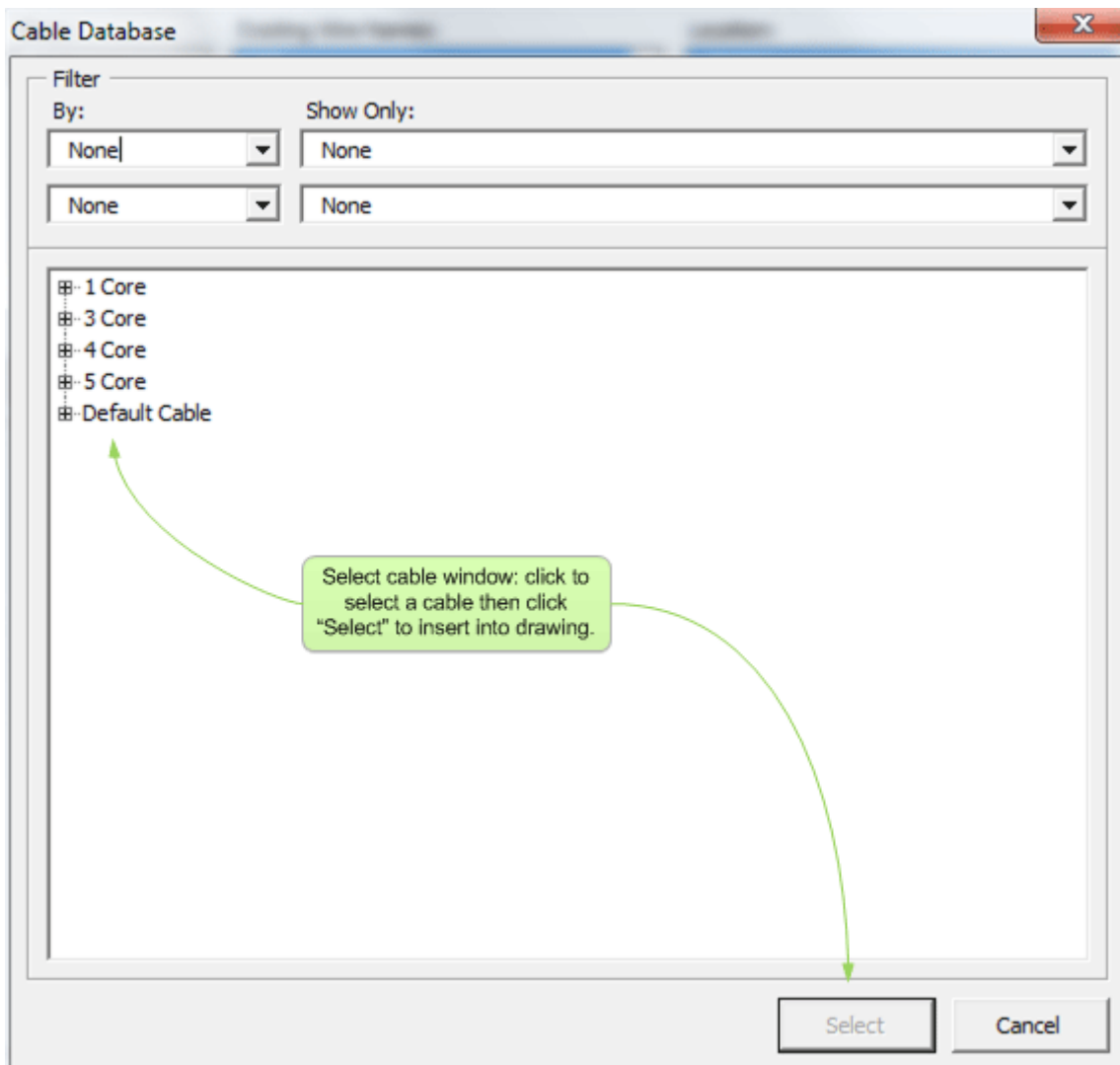
Drag and drop wires to arrange. Drag to above to remove. Drag from above to left to insert.

OK

Cancel

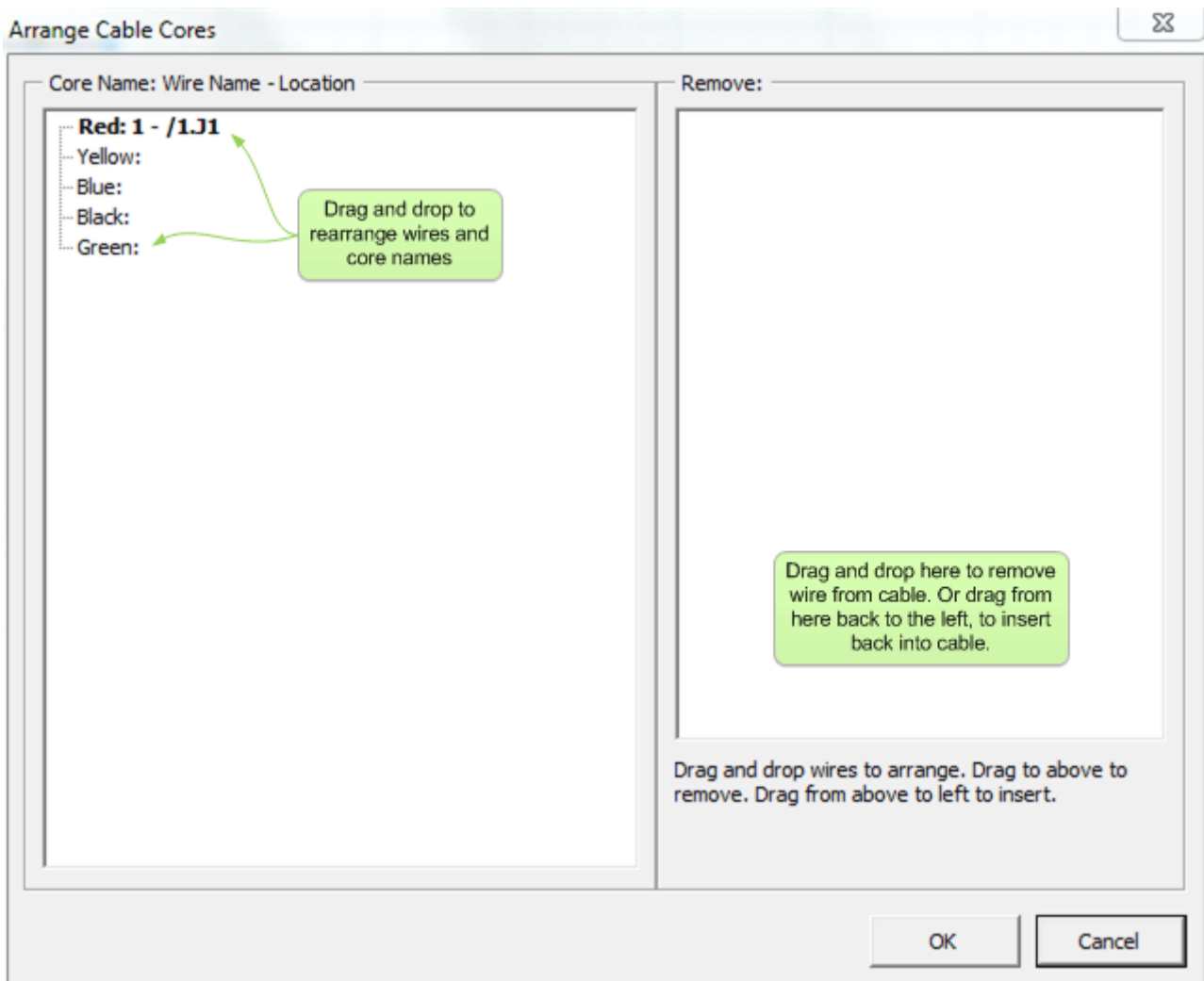
5.13 Selecting A Cable

When users click the "Select" button on the Manage Cables window, the Select Cable window will be displayed:

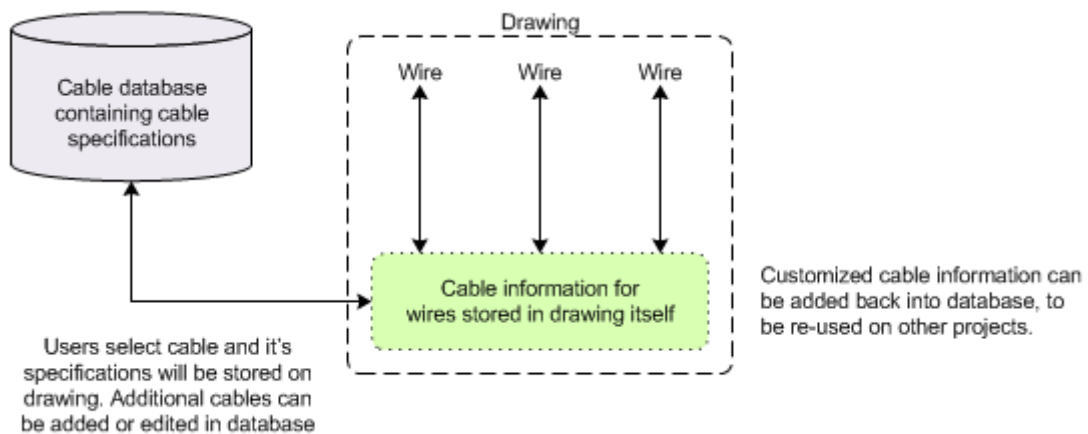


5.14 Arranging Cable Cores

When users click the "Arrange" button on the Manage Cables window, the Arrange Cable Core window will be displayed:

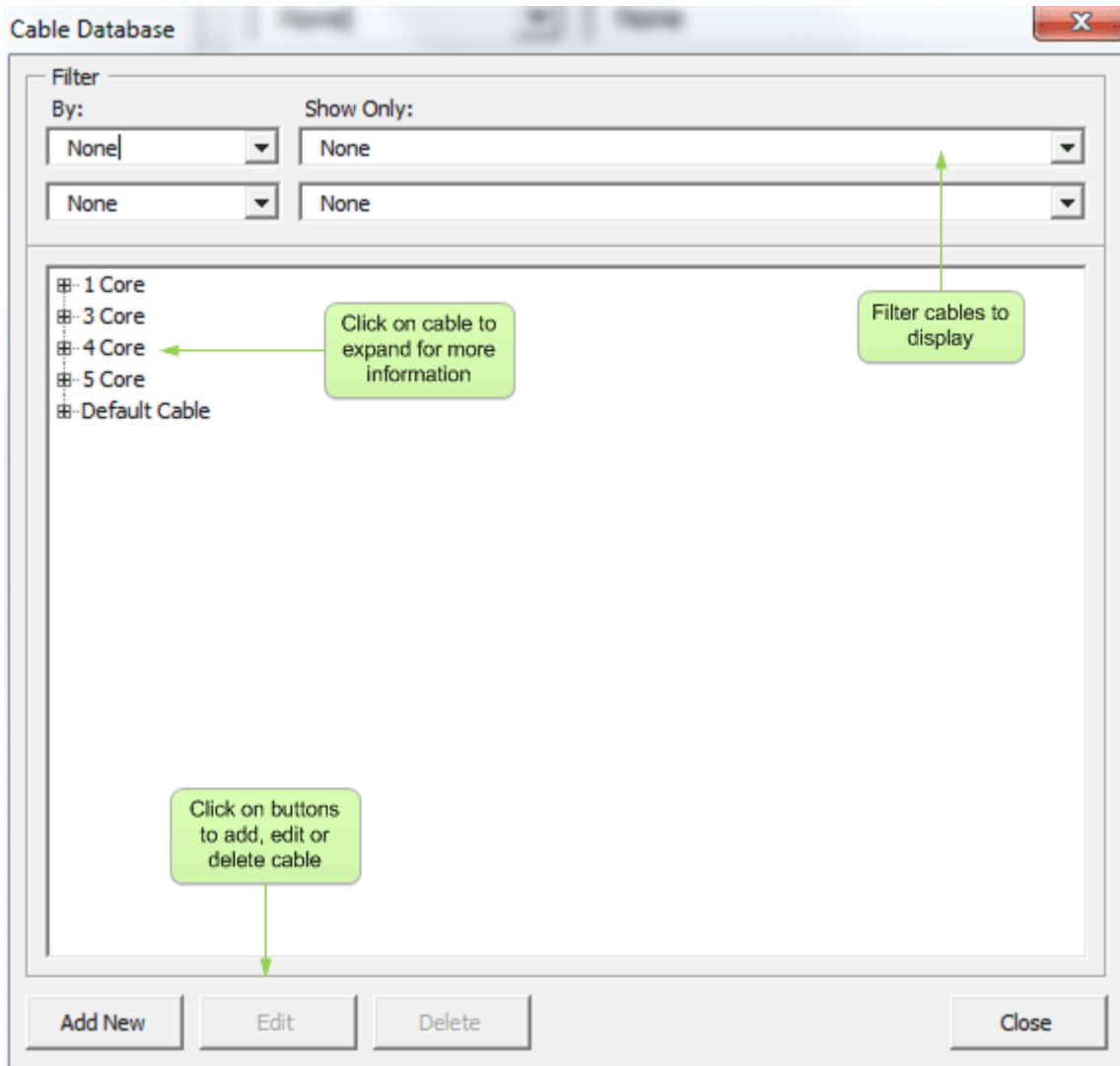


5.15 The Cable Database



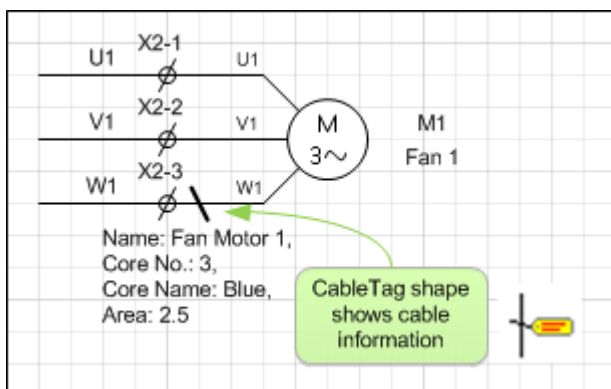
To access the cable database:

- Click on menu Electra | Cable Database



The Cable database stores all cable specifications so that they can be used and re-used on multiple projects and drawings. Once a cable is selected, its specifications are then transferred and stored on the drawing itself for portability.

5.16 The Cable Tag Shape



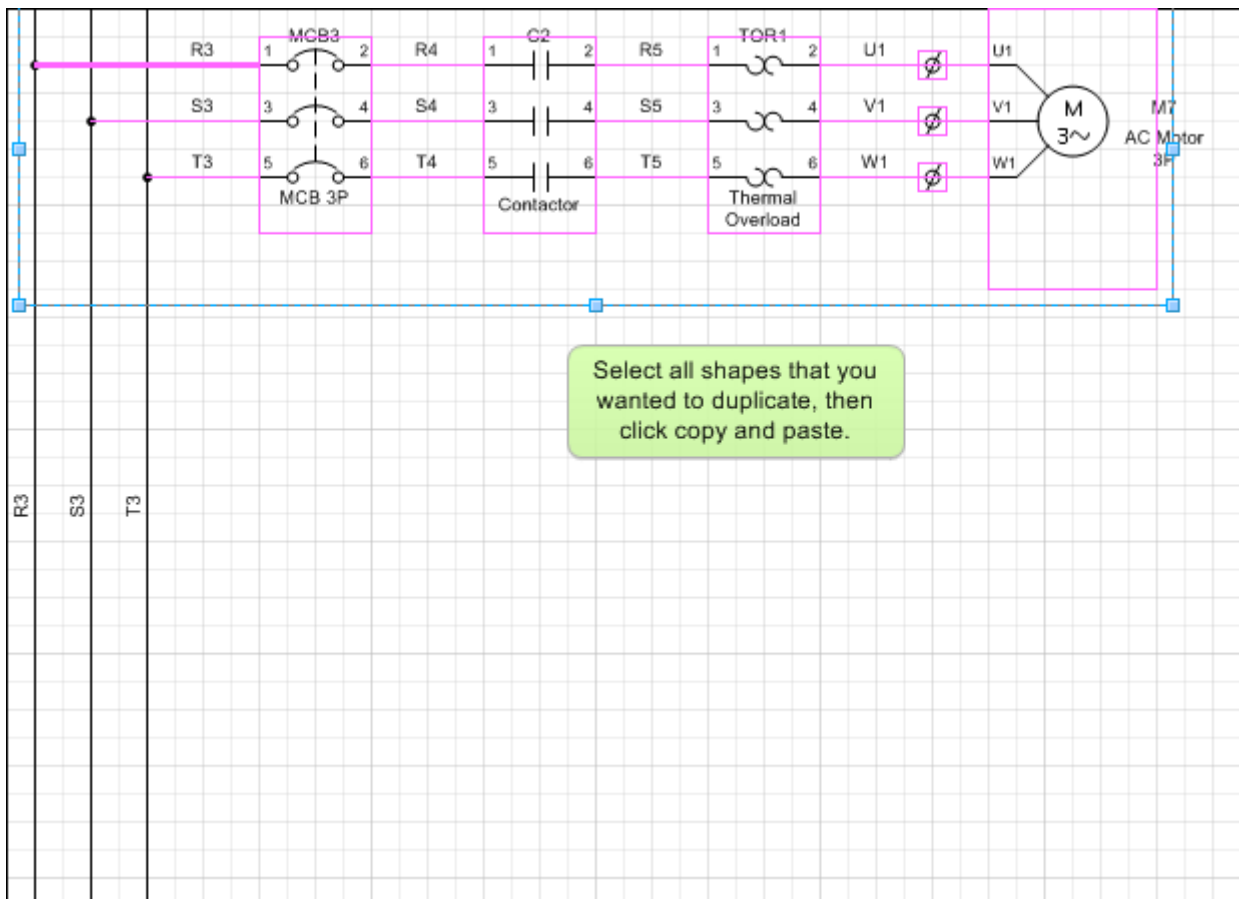
The Cable Tag shape is used on a drawing to automatically display cable information for a wire. Drag and drop the cable tag on a wire and it will automatically display cable information for that wire. Right click on the cable tag for more options.

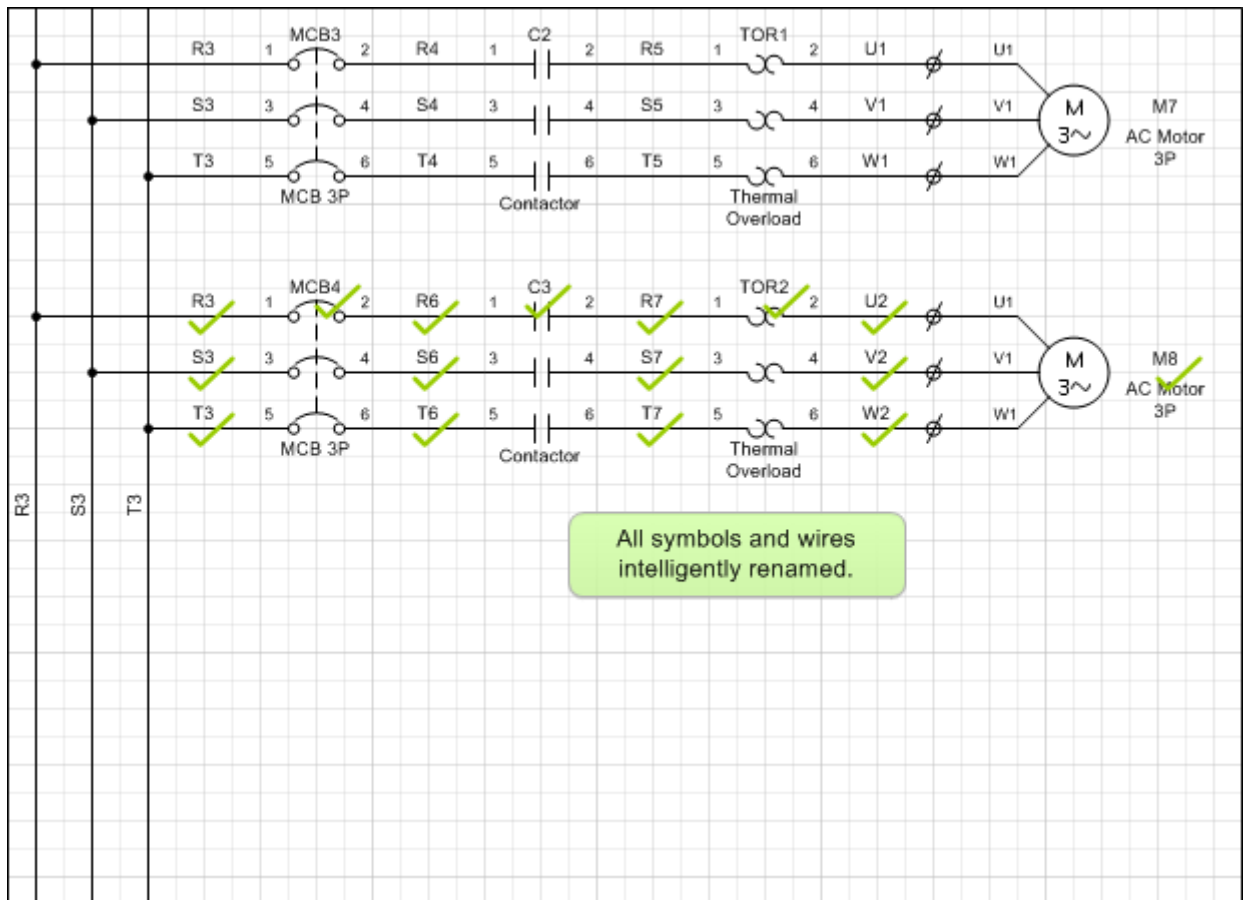
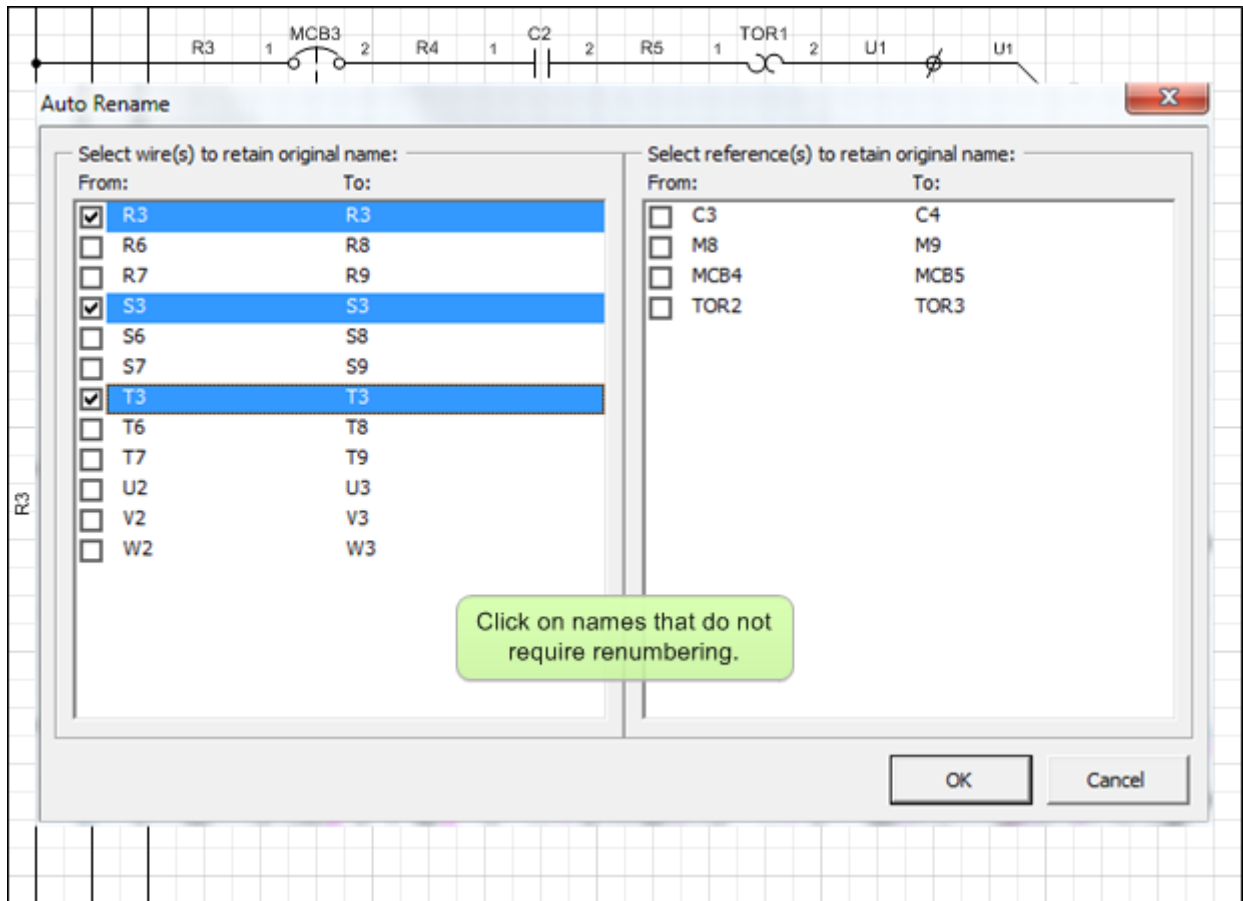
6. Re-Using Circuits

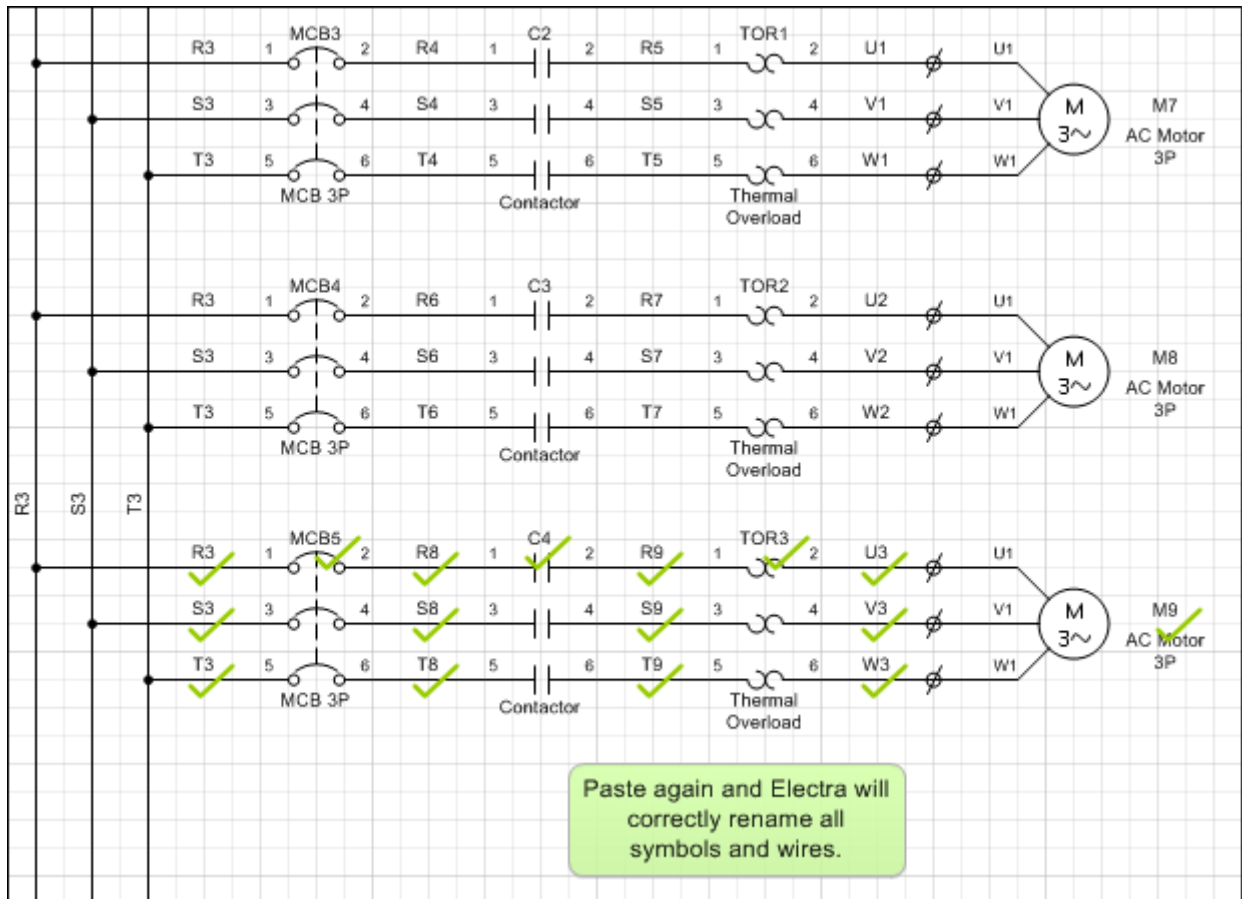
6.1 Intelligent Copy and Paste

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 wires on these circuits must be correctly and uniquely numbered. The issue is further complicated by the fact that certain symbols should be be 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 users to choose wires and references that should be left unchanged and automatically and intelligently renumbering wires and references through its AutoRename window.



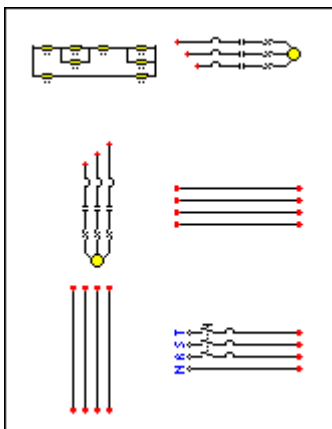




To duplicate and reuse your circuits:

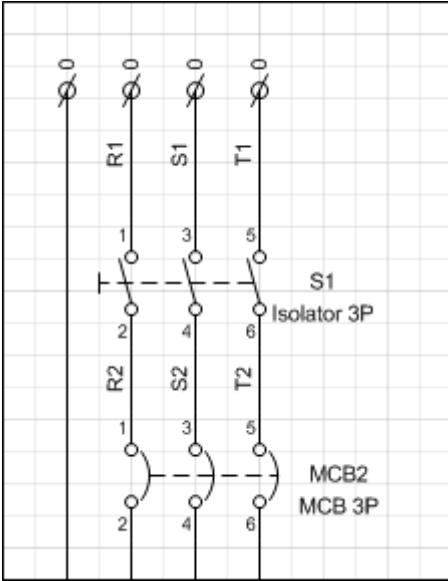
- Select all the shapes in your circuit
- Click on the copy tool then click on the paste tool
- Select/Deselect symbols and wires to be renamed on the AutoRename window
- Click OK and then drag your new circuit to your desired location

6.2 Prefab Circuits



Pre-Fabricated (Prefab) circuits are circuits that is created and stored on a stencil to be easily re-used through drag and drop. Prefab circuits are extremely portable, and easily shared among different projects, teams or even online, simply by sharing the stencil on which it resides. Users basically make a circuit, store it in a stencil and re-use with minimal work and maximum portability.

6.3 Creating and Using Prefab Circuits



To create a Prefab circuit:

- Draw a circuit as you would normally draw them
- Select all the shapes on your circuit
- Click on menu Electra | Create Prefab Circuit
- Type in a name and description for your circuit
- Select the target stencil to store the circuit
- Click OK

Your newly created prefab circuit will be then be stored in the stencil of your choosing, ready to be dragged and dropped onto your drawings.

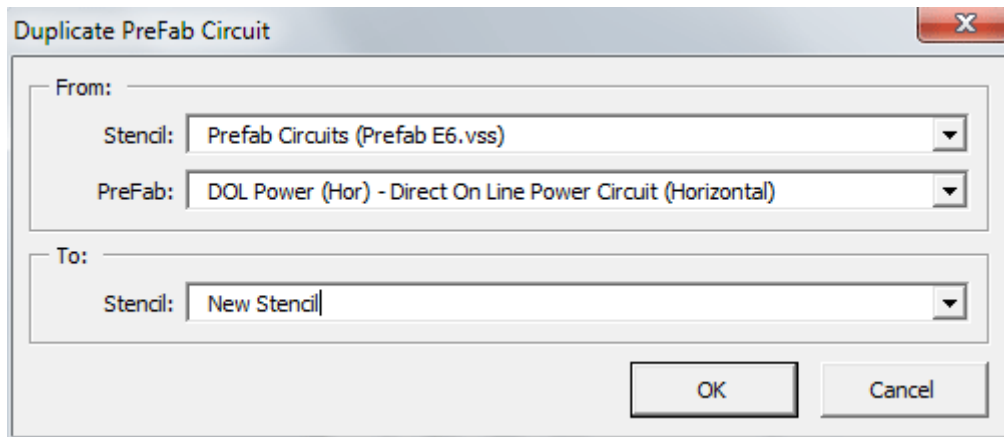
6.4 Sharing Prefab Circuits

To share a prefab circuit, simply copy the stencil with the prefab circuit and make it available on a network drive to be shared among team members, or compress the stencil to be emailed to anyone.

6.5 Moving and Duplicating Prefab Circuits

Prefab circuits CANNOT be dragged and drop to another stencil. To move or duplicate Prefab circuits:

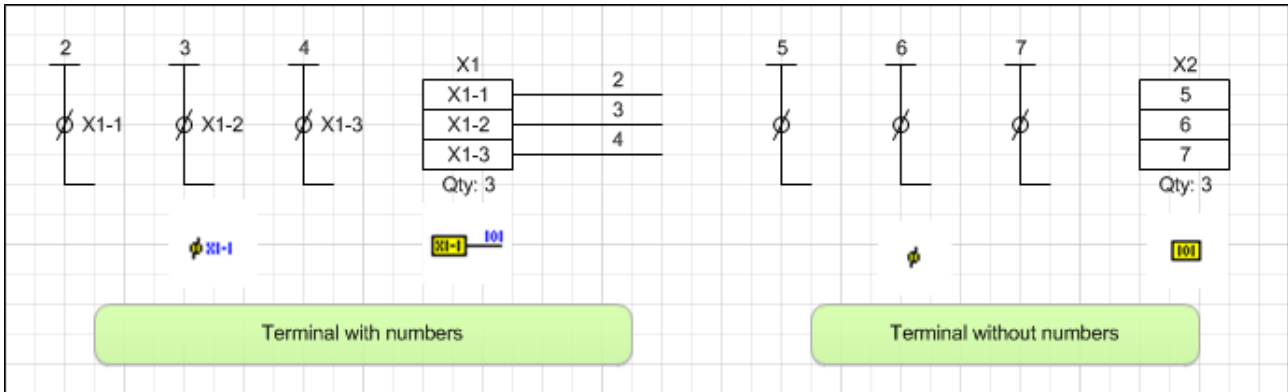
- Select menu Electra | Duplicate Prefab Circuit
- Select the source stencil in which the Prefab circuit resides
- Select "All" to duplicate all prefab circuits or select a prefab circuit from the drop down
- Select the target stencil to store the prefab circuit
- Click OK



Your Prefab circuit will now be duplicated to the target stencil of your choosing. The original prefab circuit can now be deleted simply by click on the prefab circuit on the stencil and pressing the DEL key.

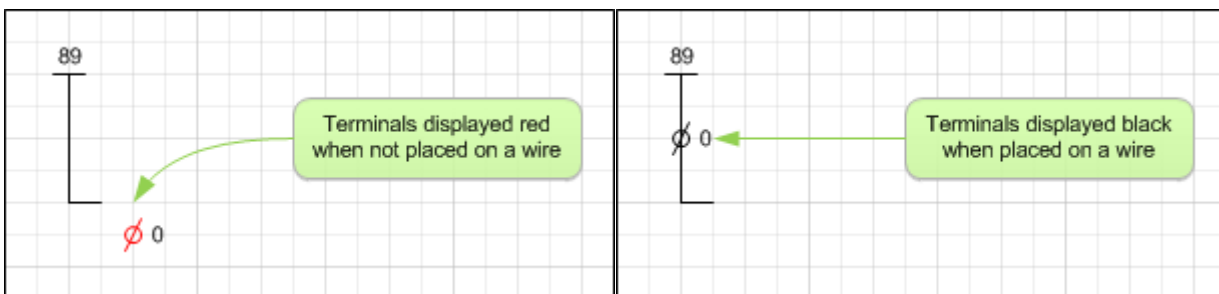
7. Intelligent Terminals

7.1 Terminal Types



There are 2 types of terminal in Electra, terminal without numbers and terminal with numbers. Terminal with numbers will show terminal number when their terminal listing is generated while terminal without numbers will not show numbers during terminal list generation.

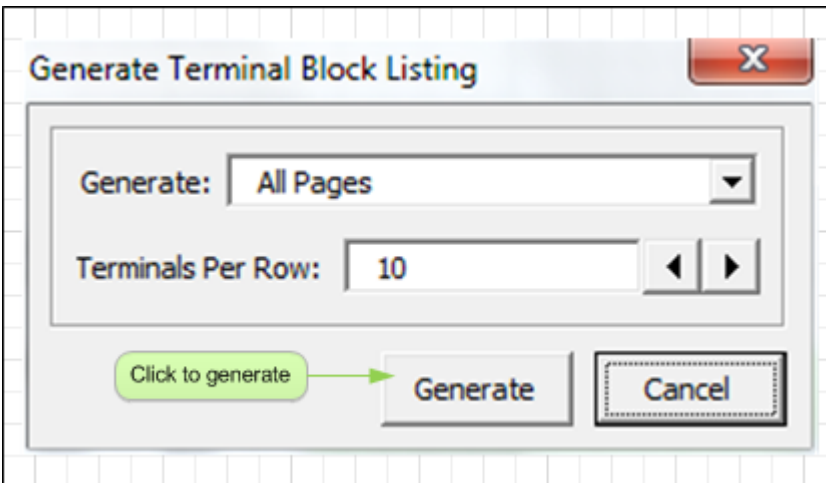
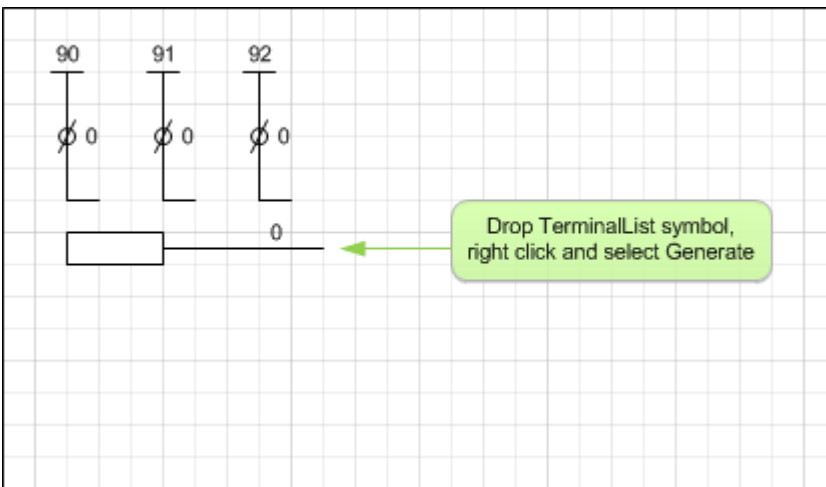
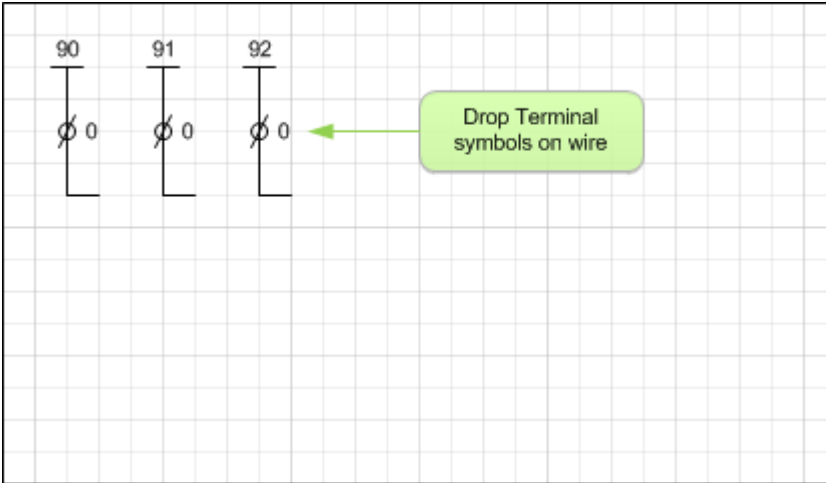
7.2 Terminal Status and Wire Detection



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

To place a terminal, drag a wire onto the drawing and then drop a terminal on the wire. When a terminal symbol is dropped on a wire, the terminal symbol will automatically follow a wire like a magnet when the wire is moved or resized. To remove this behavior, drag and drop the terminal symbol away from a wire.

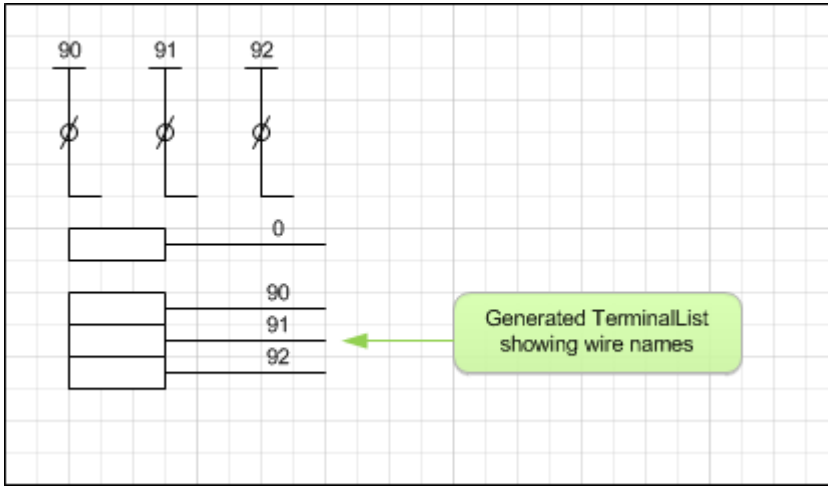
7.3 Generating Terminal Listing



To generate terminal listing with numbers use the TerminalList with number. To generate terminal listing without numbers, use TerminalList without numbers. After placing all required terminals on your drawing, generate a terminal list by:

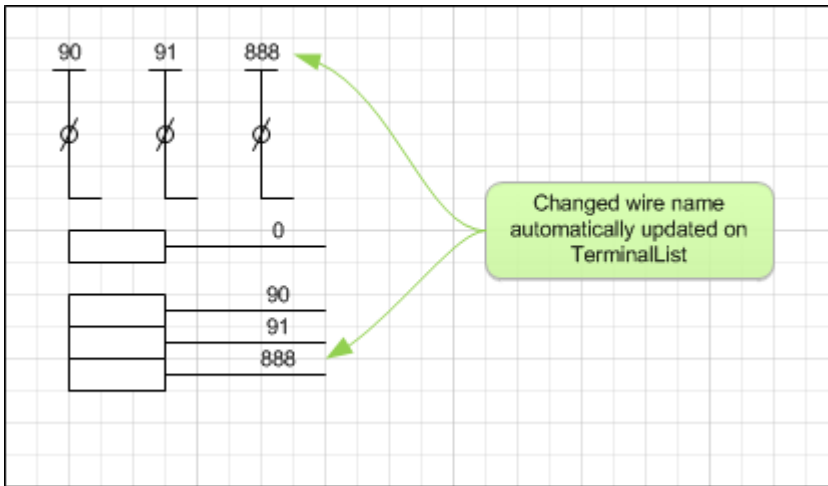
- Drag and place a TerminalList shape from the stencil
- Right click on the TerminalList shape, select Generate Listing
- Select Generate for all pages or this page only
- Click Generate

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. When users modify a wire name with a terminal on it, the wire name is automatically updated on the terminal listing.

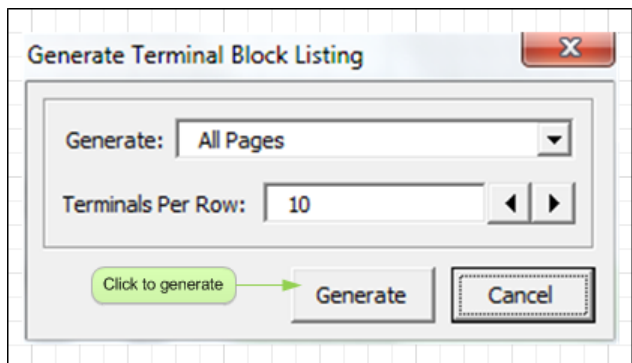
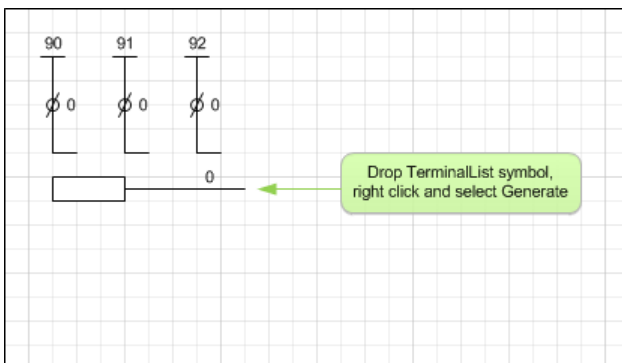
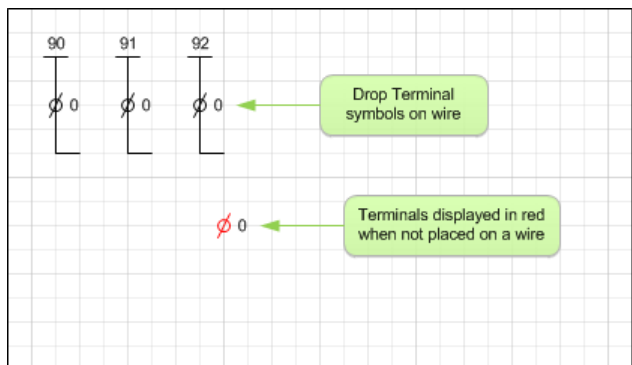
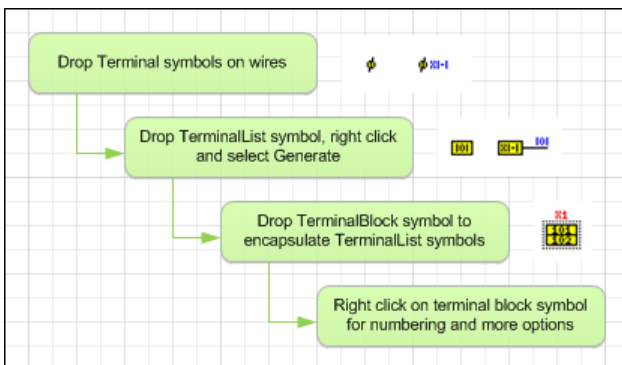


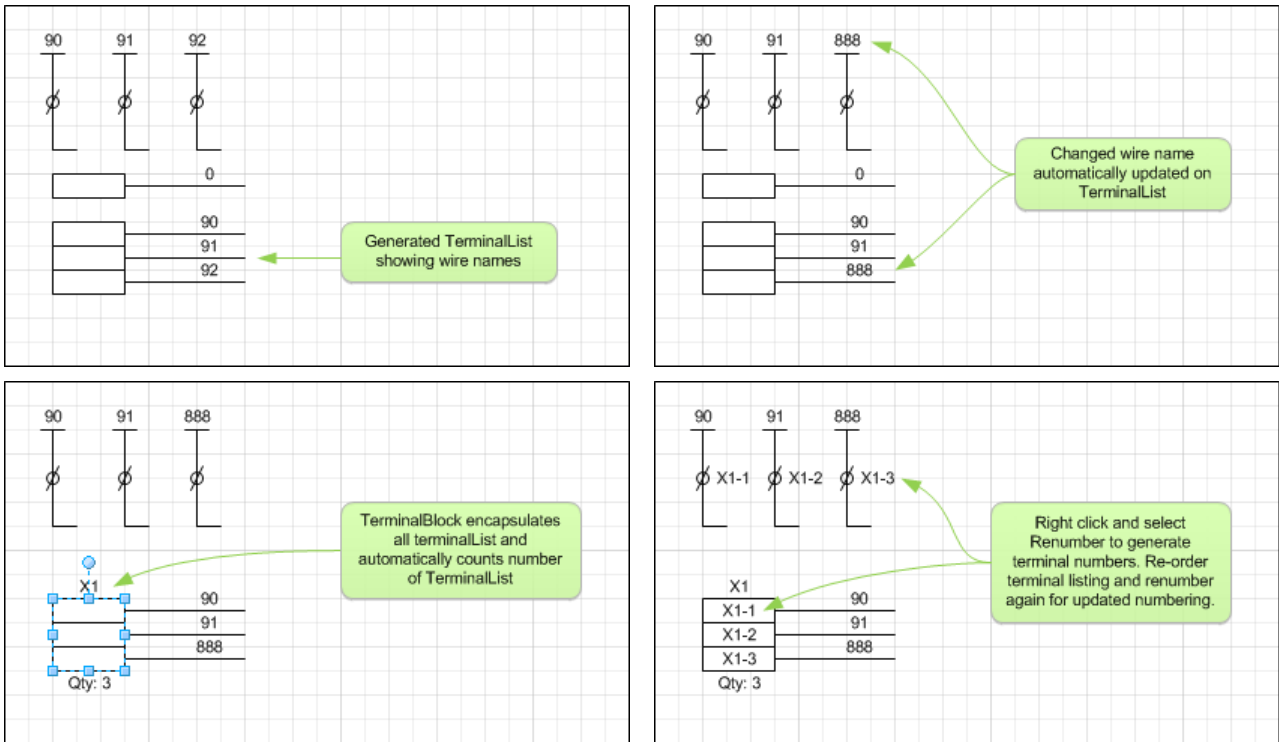
Right click on any terminal listing to jump to the terminal symbol from which the terminal listing is generated. On the terminal symbol, right click to jump to the terminal listing.

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".



7.4 Using Terminal Blocks





Terminals are normally handled as blocks. Users 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:

- Drag a TerminalBlock shape onto your drawing
- Drag and resize the TerminalBlock shape and ensure that it encapsulate all your TerminalList shape.
- Right click and select "ReNumber Terminals" to generate TerminalBlock with numbers

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, right click on the TerminalBlock shape and select "Edit Terminal Block". Fields and meanings of the Edit Terminal Block as below:

Fields	Default Value	Description
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

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.

8. Panel Layout

8.1 The Panel Layout Process

To generate panel layout, follow the process as listed below:

- Complete you schematic drawings
- Create a new page and set scale and measurement units
- Place base plates, cable ducts and rails on the new page
- Click on menu Electra | Generate Layout and use the Generate Layout window to assign components
- Click on Generate or Generate All button to generate layout
- Use guides to place and align layout shapes

8.2 Preparing a Page for Panel Layout

Pages on the New Electra drawing 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:

- Right click on a page tab and select insert page
- On the Page Setup dialog, select the Page Properties tab
- On the measurement units drop down, select millimeters
- On the Page Setup dialog, select the Drawing Scale tab
- Click on the custom scale radio button and type in your custom scale

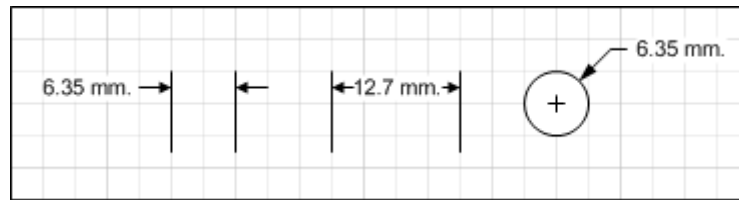
A scale of 1:5 is recommended for medium sized panels while a scale of 1:10 is recommended for larger panels.

See Also: Page Scale, Page Measurement Units

8.3 Inserting Plates, Cable Ducts And Rails

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 by right clicking on the plate. Users can also set units for plates, cable ducts and rails to be displayed on a drawing by right clicking on these shapes.

8.4 Dimension Shapes

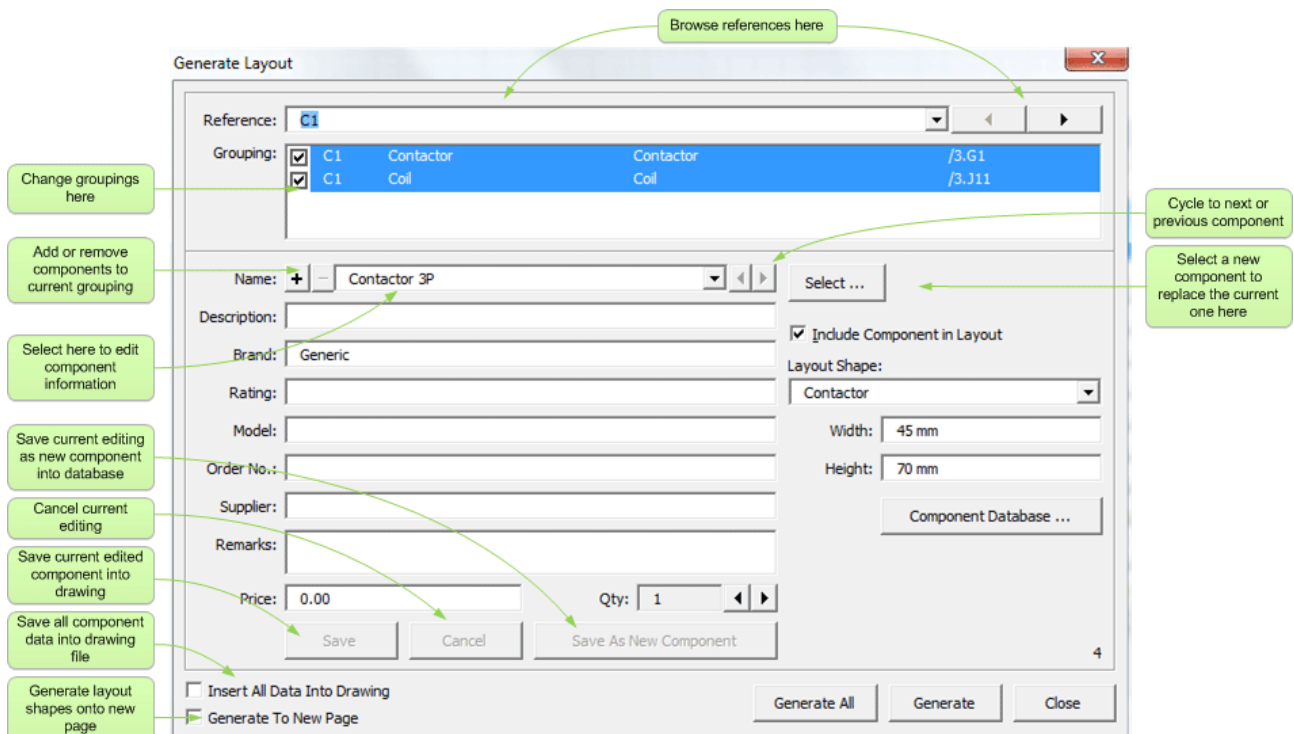


To measure length and diameters, Electra provides dimensioning shapes on the Layout stencil to be dragged and dropped on drawings. All of them automatically display the dimension when their handles are dragged. Right click on dimension shapes for more options.

8.5 Generating Panel Layout

To generate panel layout:

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



Every time you run Generate Layout and click the Generate All button, Electra will automatically synchronize your drawings. For example, you may have added or deleted some symbols on your schematics, and Generate Layout will add in or delete components accordingly. If you have changed the references of your schematics, GenerateLayout will also automatically reflect this change in your layout.

See Also: Understanding Symbols and Components, How Symbols are Matched to Components

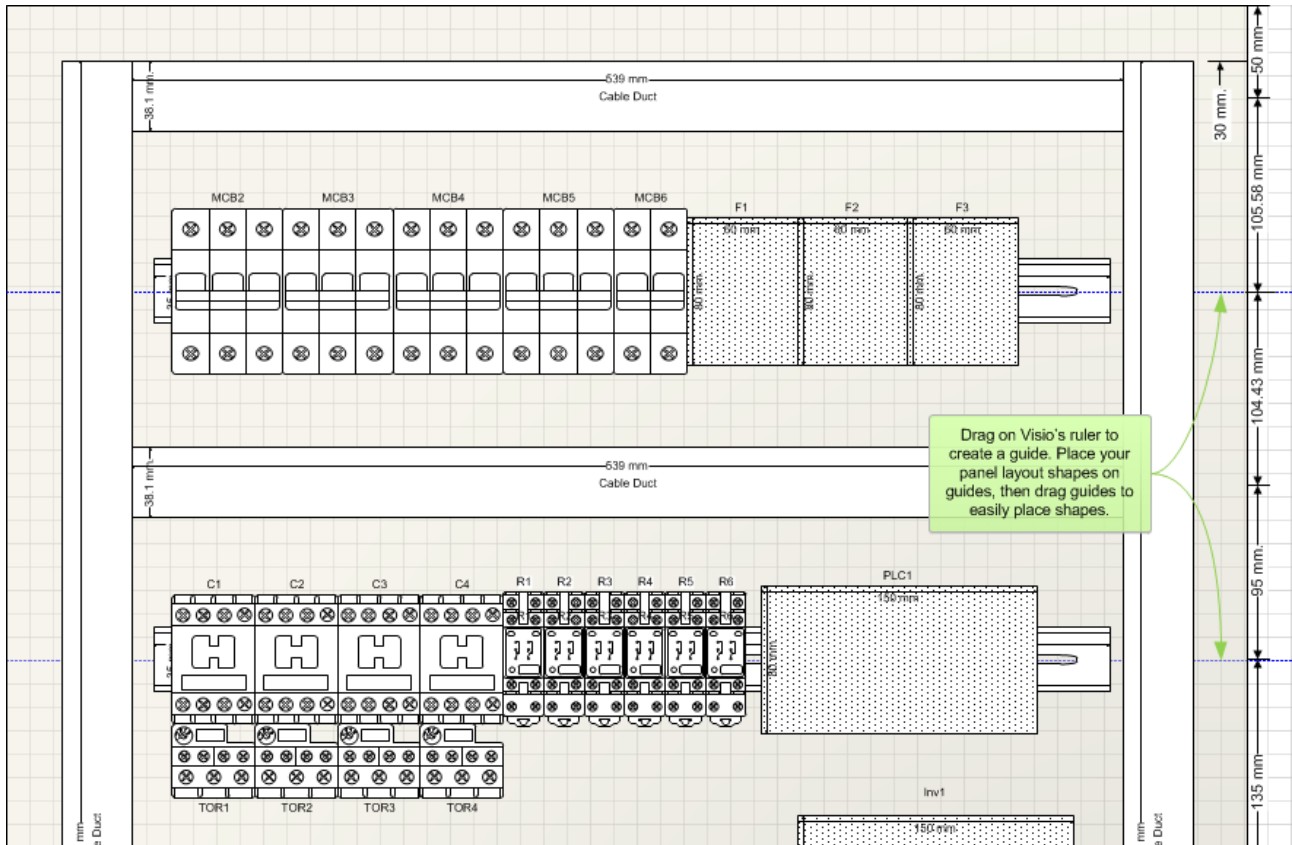
8.6 Using Guides to Place Layout Shapes

To create a guide:

- Drag on the ruler in Visio

Placing shapes on a guide:

- Drag shapes onto the guide (center of shape will turn red to indicate latching on guide)



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

8.7 Using 3D Layout Shapes

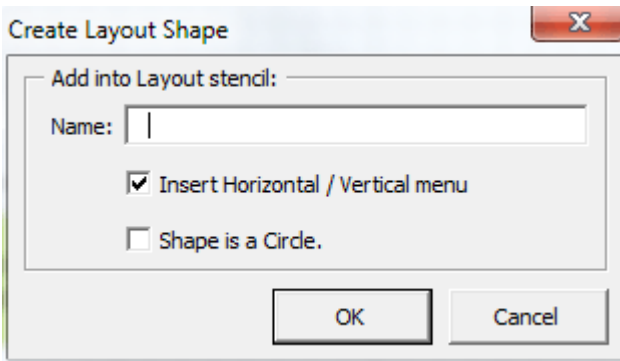


To use the Layout 3D shapes, open the stencil "Layout3D E6.vss" in the C:\Program Files\Radica\Electra\ folder. Make sure the folder is opened in your document when you generate layout.

8.8 How Symbols are Matched to Components

- All symbols in a drawing are divided into references
- Symbols with the same references are grouped together and can be divided into further groups
- Each group of symbols can be assigned to one or more components
- For each group of symbols, Electra looks for the symbol with the lowest priority to obtain the default component
- The default component can then be changed to any other component by users using the menu Electra | Manage Components

8.9 Making Your Own Layout Shapes



To make your own layout shape:

- Draw lines, arcs and graphics or
- Import a CAD drawing and convert into Visio or
- Import an image of the layout shape
- Group all shapes and select them
- Click on menu Electra | Create Layout Shape
- Type in a name and description for the layout shape
- Click OK

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 (In Visio 2010, it's called preview).

9. Generating Reports

All reports that is generated in drawing automatically creates another report shape when a report overflows. This enables users to place reports over multiple pages when reports are long and cannot fit into a single page.

9.1 Cross Reference Report

Reference	Description	Master	Location
C1	NO Contact	Normally Open Contact	/4.I2
C1	Coil	Coil	/4.I6
C1	Contact	Contact	/2.G4
C2	NO Contact	Normally Open Contact	/4.G2
C2	Coil	Coil	/4.G6
C2	Contact	Contact	/2.E4
C3	NO Contact	Normally Open Contact	/4.E2
C3	Coil	Coil	/4.E6
C3	Contact	Contact	/2.C4
C4	NO Contact	Normally Open Contact	/4.C4
C4	Coil	Coil	/4.C6
C4	Contact	Contact	/2.I11
ES1	Emergency Stop	Emergency Stop NC	/3.C5
ES2	Emergency Stop	Emergency Stop NC	/3.C6
F1	Fuse 2P	Fuse 2P	/3.I2
F2	Fuse 2P	Fuse 2P	/3.G2
F3	Fuse 2P	Fuse 2P	/3.E2
G1	Power Supply	Power Supply, Rectifier - AC/DC	/3.B8
H1	Heater 1	Heater	/3.I5
H2	Heater 2	Heater	/3.G5
H3	Heater 3	Heater	/3.E5
ISO1	Isolator 3P	Isolator 3P	/2.I1
Inv1	Inverter	Inverter	/2.D10

To generate a Cross Reference report in a drawing:

- Drop the Cross Reference report shape onto your drawing
- Resize the report shape as required by dragging on the handles
- Right click and select Generate
- After generation, click once on the report shape, then click again on a column and resize the column by dragging on the handles

To export the Cross Reference report, click on menu Electra | Export | Cross Reference Report

9.2 Material Reference Report

Reference	Component Name	Description	Brand	Rating	Model	Quantity
C1	Contactora 3P		Generic			1
C2	Contactora 3P		Generic			1
C3	Contactora 3P		Generic			1
C4	Contactora 3P		Generic			1
ES1	Emergency Stop - Turn Reset	Push lock, Turn Reset	Generic			1
ES2	Emergency Stop - Turn Reset	Push lock, Turn Reset	Generic			1
F1	Fuse 2P		Generic			1
F2	Fuse 2P		Generic			1
F3	Fuse 2P		Generic			1
G1	Power Supply AC/DC - Switching	Switching Power Supply	Generic			1
H1	Heater	Heater	Generic			1
H2	Heater	Heater	Generic			1
H3	Heater	Heater	Generic			1
ISO1	Isolator/Disconnectora 3P	Panel Mount	Generic			1
Inv1	Inverter	Motor Inverter	Generic			1
M1	Motor 3P AC		Generic			1
M2	Motor 3P AC		Generic			1
M3	Motor 3P AC		Generic			1
M4	Motor 3P AC		Generic			1
M5	Motor 3P AC		Generic			1
MCB1	MCB 3P	Miniature Circuit Breaker	Generic			1
MCB2	MCB 3P	Miniature Circuit Breaker	Generic			1
MCB3	MCB 3P	Miniature Circuit Breaker	Generic			1

To generate a Material Reference report in a drawing:

- Drop the Material Reference report shape onto your drawing
- Resize the report shape as required by dragging on the handles
- Right click and select Generate
- After generation, click once on the report shape, then click again on a column and resize the column by dragging on the handles

To export the Material Reference report, click on menu **Electra | Export | Material Reference Report**

9.3 Bills of Materials Report

Component Name	Description	Brand	Rating	Model	Quantity	Price	Amount
Cable Duct	(W x H) 332.0000 mm x 38.1000 mm	Generic			1	0.00	0.00
Cable Duct	(W x H) 539.0000 mm x 38.1000 mm	Generic			4	0.00	0.00
Cable Duct	(W x H) 850.0000 mm x 38.1000 mm	Generic			2	0.00	0.00
Clutch		Generic			1	0.00	0.00
Contacto 3P		Generic			4	0.00	0.00
DIN Rail	(W x H) 320.0000 mm x 35.0000 mm	Generic			1	0.00	0.00
DIN Rail	(W x H) 520.0000 mm x 35.0000 mm	Generic			3	0.00	0.00
DIN Rail	(W x H) 615.0000 mm x 35.0000 mm	Generic			1	0.00	0.00
Emergency Stop - Turn Reset	Push lock, Turn Reset	Generic			2	0.00	0.00
Fuse 2P		Generic			3	0.00	0.00
Heater	Heater	Generic			3	0.00	0.00
Inverter	Motor Inverter	Generic			1	0.00	0.00
Isolator/Disconnecto 3P	Panel Mount	Generic			1	0.00	0.00
MCB 2P	Miniature Circuit Breaker Double Pole	Generic			1	0.00	0.00
MCB 3P	Miniature Circuit Breaker	Generic			5	0.00	0.00
Motor 3P AC		Generic			5	0.00	0.00
PLC Module		Generic			1	0.00	0.00
Photoelectric Sensor - Through Beam, Both	Through Beam, Transmitter & Receiver Set	Generic			1	0.00	0.00
Pilot Light		Generic			9	0.00	0.00
Potentiometer		Generic			1	0.00	0.00
Power Supply AC/DC - Switching	Switching Power Supply	Generic			1	0.00	0.00
Proximity Switch		Generic			2	0.00	0.00
Push Button - Momentary	Spring Return	Generic			9	0.00	0.00
Relay - MY2/MY4	MY2/MY4	Generic			6	0.00	0.00
Relay Base 2P		Generic			6	0.00	0.00
Selector Switch 2 Pos - Maintained	Maintained	Generic			3	0.00	0.00

To generate a Bills Of Material (BOM) report in a drawing:

- Drop the BOM report shape onto your drawing
- Resize the report shape as required by dragging on the handles
- Right click and select Generate
- After generation, click once on the report shape, then click again on a column and resize the column by dragging on the handles

To export the BOM report, click on menu **Electra | Export | Bills of Material Report**

9.4 Cable Reference Report

Cable	Core Name	Wire	Location
Name: Fan Motor 1	Red	U1	/2.G6
Brand: Generic	Yellow	V1	/2.G6
Model: Generic	Blue	W1	/2.F6
Cores: 3			
Area: 2.5			
Order No.:			
Supplier:			
Remarks:			
Price: 0			
Name: Fan Motor 2	Red	U2	/2.E6
Brand: Generic	Yellow	V2	/2.E6
Model: Generic	Blue	W2	/2.D6
Cores: 3			
Area: 2.5			
Order No.:			
Supplier:			
Remarks:			
Price: 0			

To generate a Cable Reference report in a drawing:

- Drop the Cable Reference report shape onto your drawing
- Resize the report shape as required by dragging on the handles
- Right click and select Generate
- After generation, click once on the report shape, then click again on a column and resize the column by dragging on the handles

To export the Cable Reference report, click on menu Electra | Export | Cable Reference Report

9.5 Connections (From/To) Report

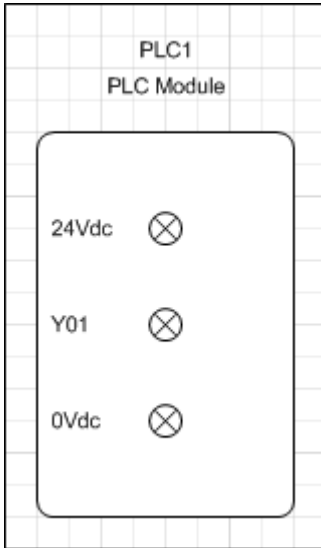
Wire	Location	Cable	Core No.	Core Name	Reference	Location	Pin Name	Reference	Location	Pin Name
AI+	/2.E11				Res1	/2.E12	3	Inv1	/2.D10	AI+
AO+	/2.E12				P1	/2.D13	1	Inv1	/2.D10	AO+
AO-	/2.D12				P1	/2.D13	2	Inv1	/2.D10	AO-
N	/3.C1				MCB6	/3.C2	3	X1-1	/2.J1	
N	/3.E1				F3	/3.E2	3	X1-1	/2.J1	
N	/3.G1				F2	/3.G2	3	X1-1	/2.J1	
N	/3.I1				F1	/3.I2	3	X1-1	/2.J1	
N	/2.F1				X1-1	/2.J1				
N1	/3.I3				R1	/3.I4	23	F1	/3.I2	4
N2	/3.G3				R2	/3.G4	23	F2	/3.G2	4
N3	/3.E3				R3	/3.E4	23	F3	/3.E2	4
N4	/3.C3				SS1	/3.C4	23	MCB6	/3.C2	4
N5	/5.E13				X3-2	/3.C3				
N5	/5.E13				R6	/5.E13	A2	R5	/5.E13	A2
N5	/5.F13				R5	/5.E13	A2	R4	/5.F13	A2
N5	/5.G13				R4	/5.F13	A2	PL9	/5.G13	X2
N5	/4.A7				PL9	/5.G13	X2	PL8	/4.A6	X2
N5	/4.B7				PL8	/4.A6	X2	PL7	/4.C6	X2
N5	/4.C7				X3-48	/4.A7				
N5	/4.C7				PL7	/4.C6	X2	C4	/4.C6	A2
N5	/4.D7				X3-46	/4.C7				
N5	/4.D7				C4	/4.C6	A2	PL6	/4.D6	X2
N5	/4.F7				C3	/4.E6	A2	PL4	/4.F6	X2
N5	/4.D7				PL6	/4.D6	X2	PL5	/4.E6	X2
N5	/4.E7				X3-36	/4.D7				
N5	/4.E7				PL5	/4.E6	X2	C3	/4.E6	A2
					X2-34	/4.F7				

To generate a Connections report in a drawing:

- Drop the Connections report shape onto your drawing
- Resize the report shape as required by dragging on the handles
- Right click and select Generate
- After generation, click once on the report shape, then click again on a column and resize the column by dragging on the handles

To export the Connections report, click on menu Electra | Export | Connections Report

10. The PLC Module Shape



To create a PLC module in your drawings, drag and drop the PLC module shape onto your drawings.

To rotate the PLC module shape:

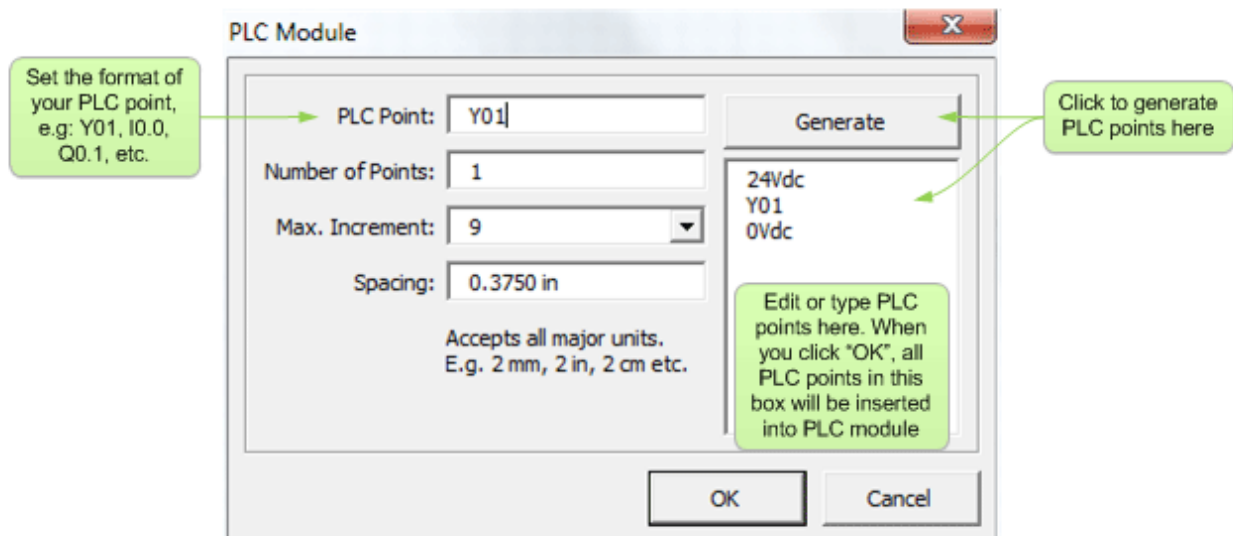
- Right click on the PLC module shape and select Horizontal or Vertical

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

- Right click on the PLC module shape and select Change Label position or Change Label orientation

10.1 Customizing the PLC Module

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



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

PLC Point	X0	B00001	Q0.0
Number of Points	8	12	12
Max. Increment	9	9	7
Original text box content	24Vdc Y01 Y02	24Vdc X0 0Vdc	24Vdc Y01 0Vdc
Results 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

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. 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.

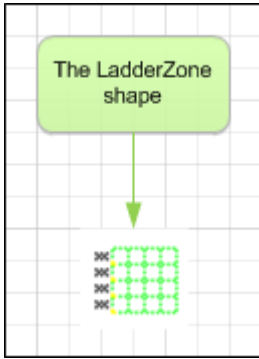
After setting PLC point, number of points and maximum increment, click on the Generate button to generate your PLC points. The generated points will be shown on the text box under the Generate button. Examples of generation at left:

Users can also manually create 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.

10.2 Creating A Single Module with Input And Output

You can use the PLC Module shape to create input and output modules and they will be displayed as separate shapes on your drawing. If you edit their reference (e.g both input and output module are "PLC1") then Electra understands that both symbols represent only a single PLC module. Therefore, only a single PLC module appears on Generate Layout or on the Bills of Materials, despite having 2 separate PLC module shape on your drawings.

11. The Ladder Zone Shape

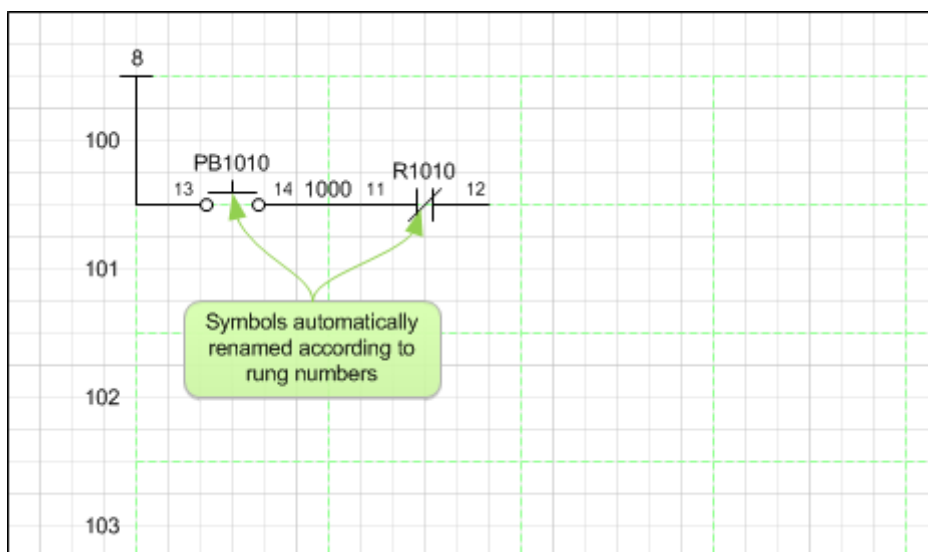
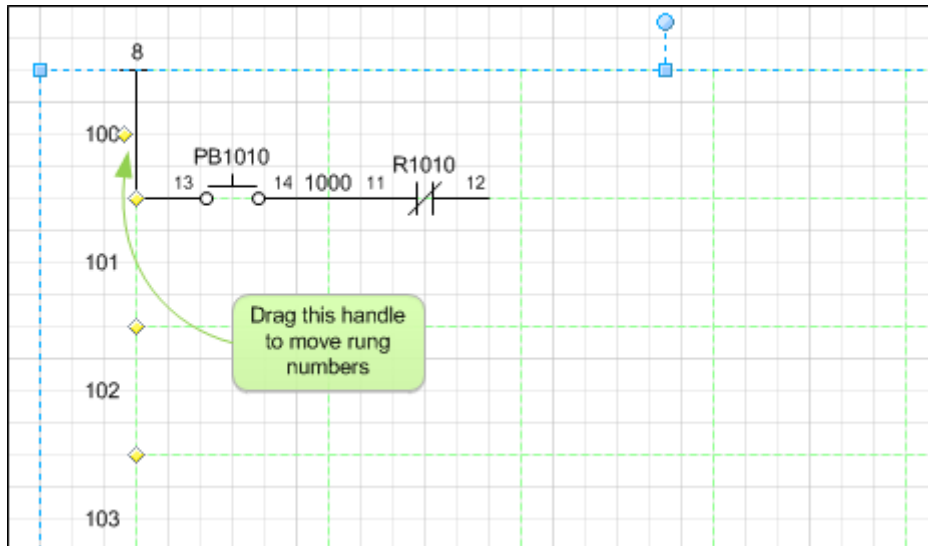


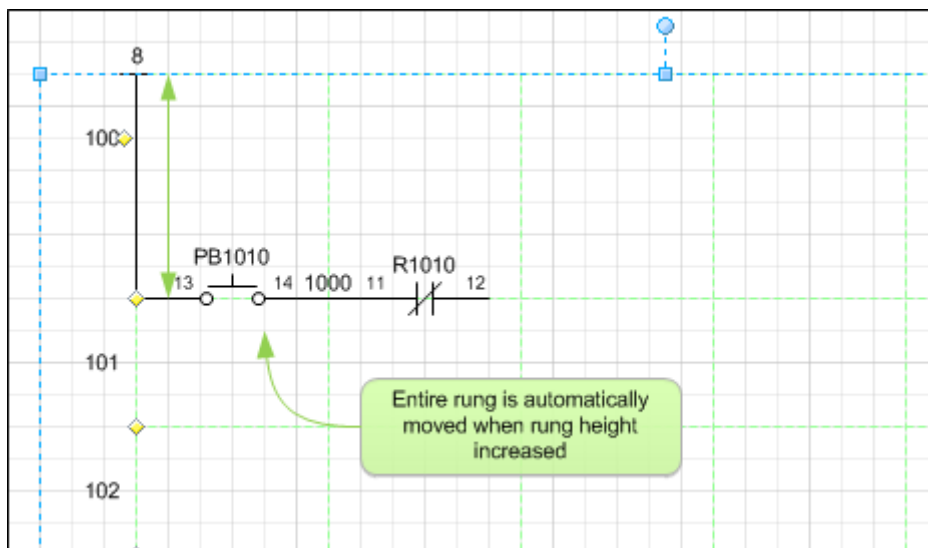
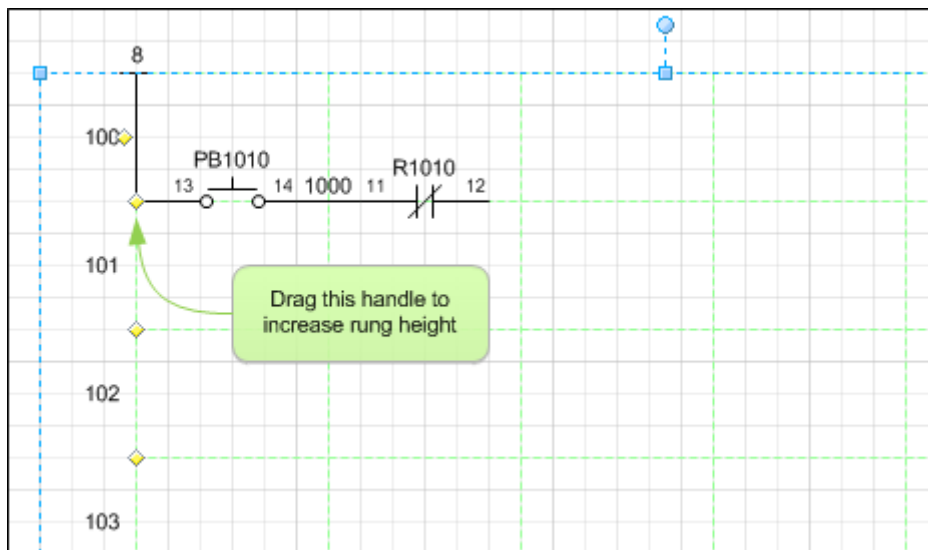
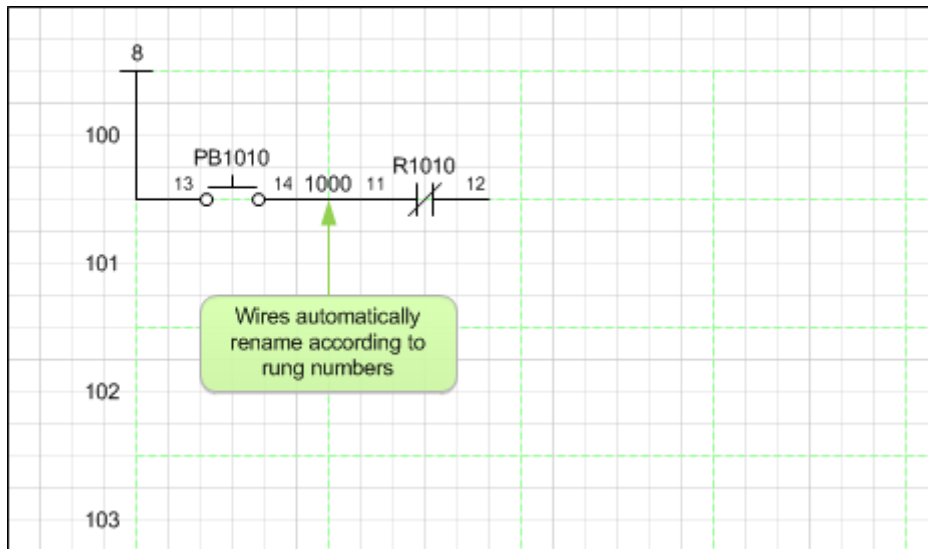
Drag and drop the LadderZone shape onto your drawings to create ladder diagrams. Symbols and wires in a ladder zone gets automatically renamed according to their rung numbers. If a number is used, it is automatically incremented to the next number. To increase the height of a rung, drag on the yellow controls handles on the LadderZone shape. All symbols attached to a rung you have moved will be automatically moved also.

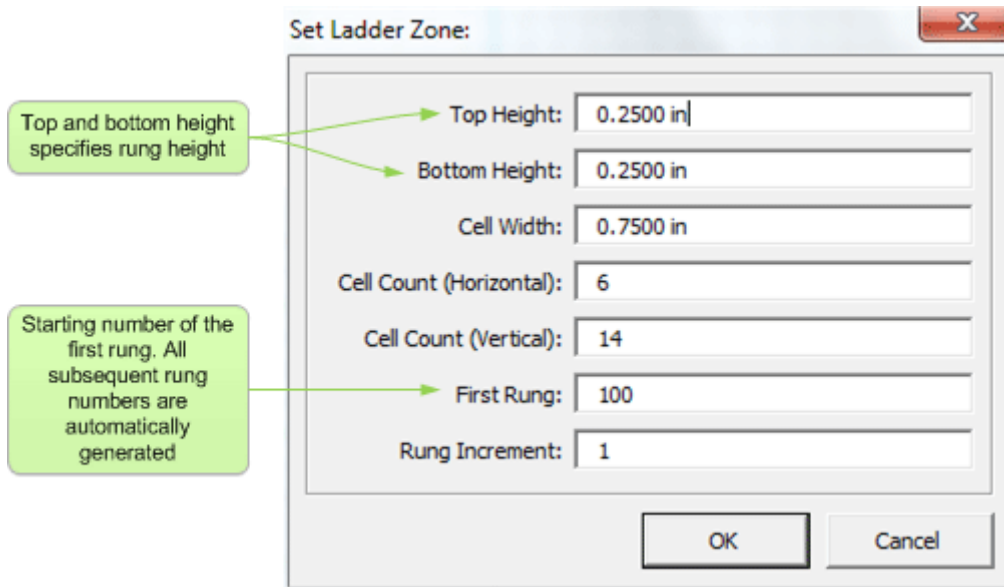
11.1 Using Ladder Zone

To use the LadderZone shape:

- Drag and drop the LadderZone shape onto your drawing







Rung numbers, and ladder zone sizing can be customized by right clicking on the LadderZone shape and selecting "Set Ladder Zone". The LadderZone shape has green grid lines that are for display only and is not printable.

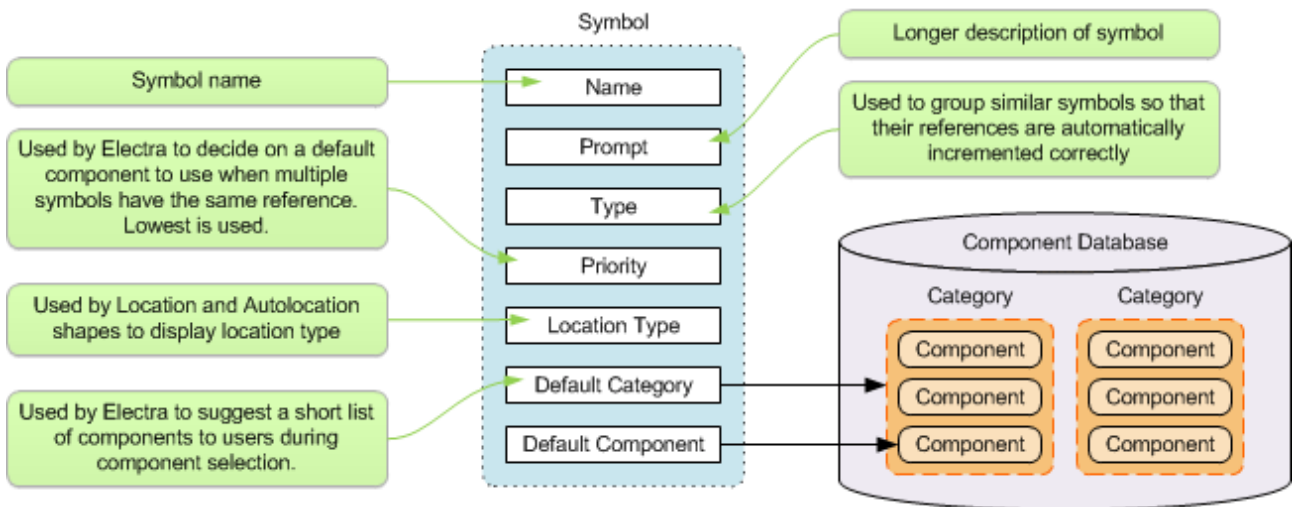
12. Creating Your Own Symbol

12.1 Symbol Creation Process

Creating your own intelligent schematic symbol is an easy and automated process:

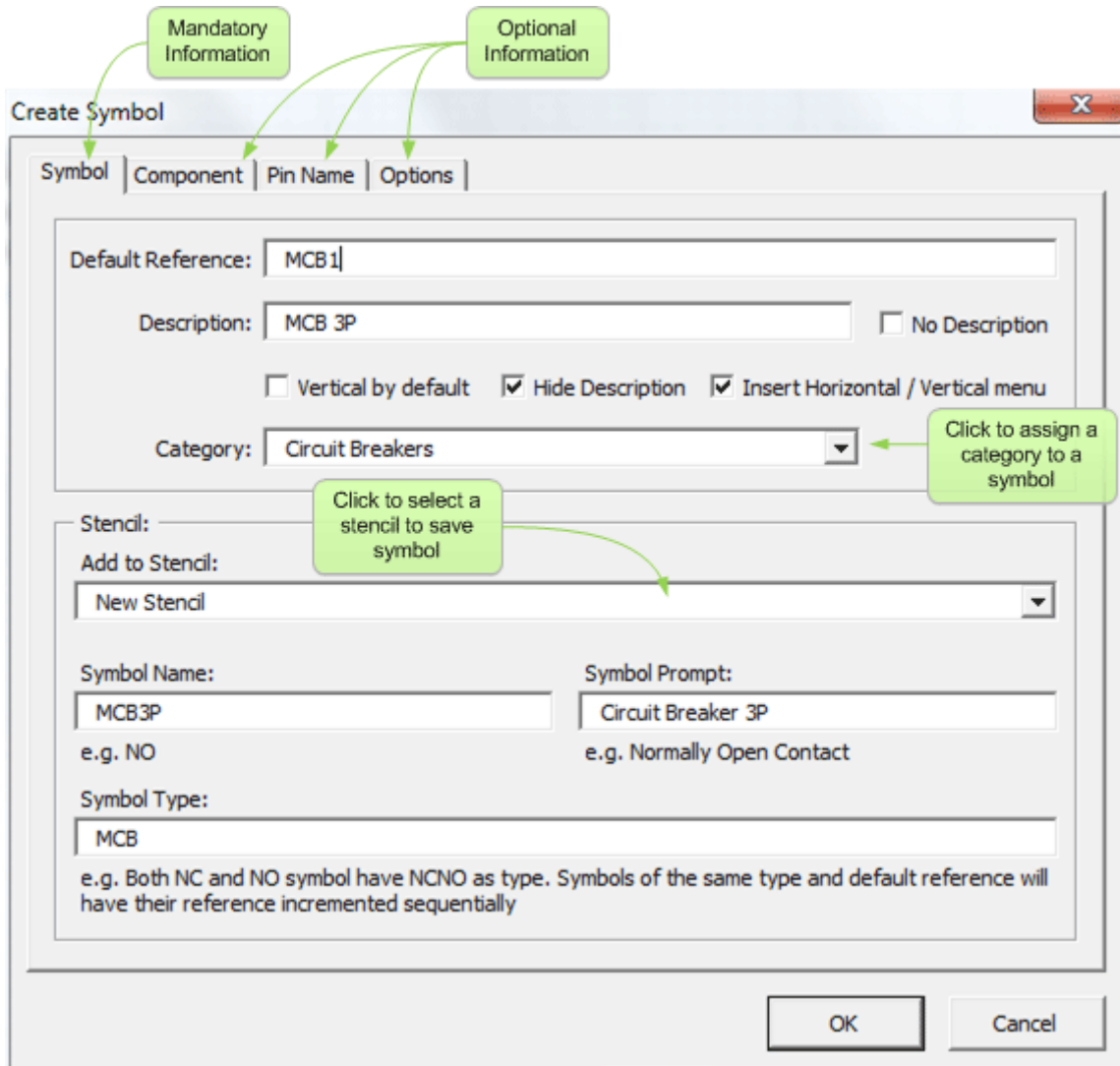
- Draw symbols using lines, circles, arcs and text
- Group all shapes and add connection points
- Select shape and click on menu Electra | Create Schematic Symbol

12.2 The Anatomy of a Symbol

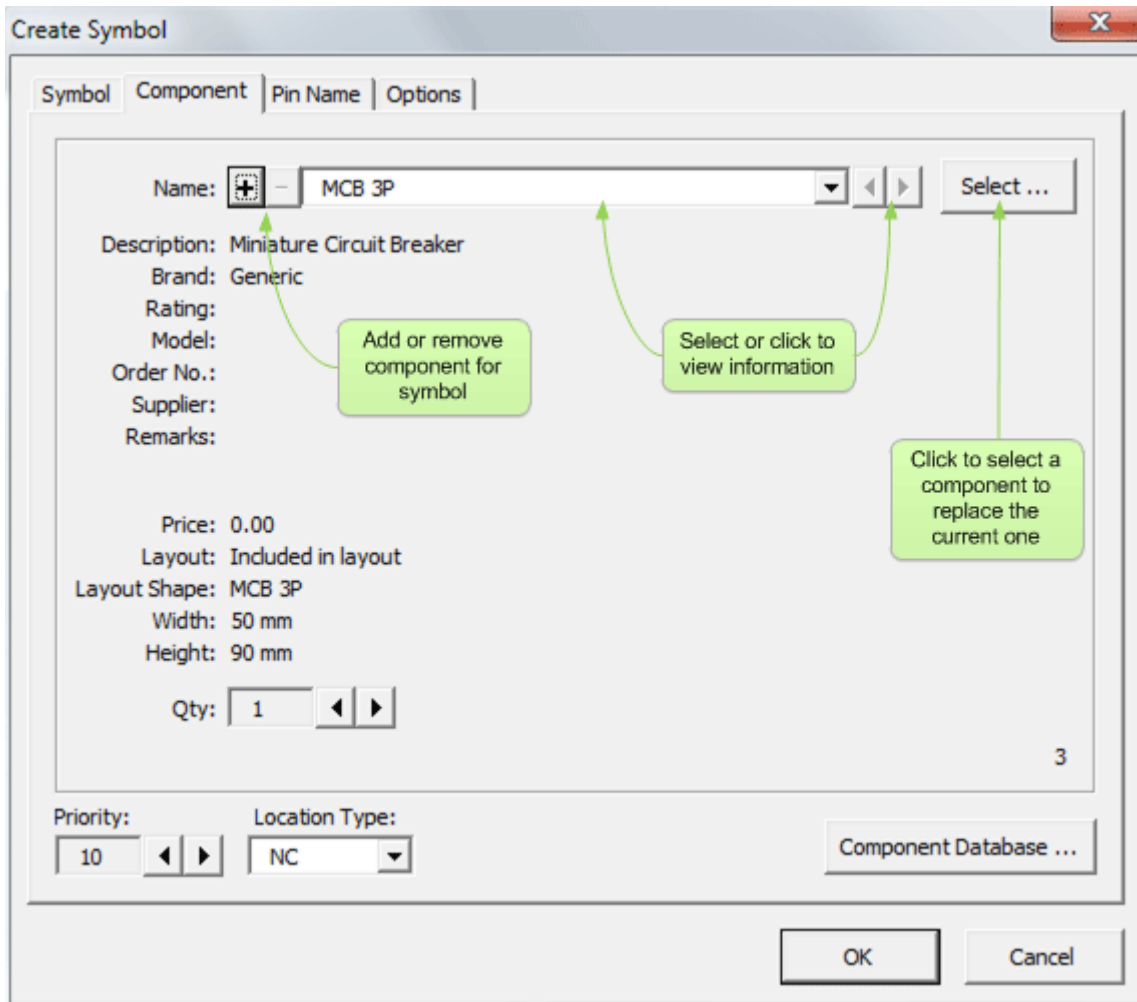


12.3 The Create Symbol Window

The Create Symbol 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.



Default Reference	Default reference for your custom symbol (e.g. R1)
Description text box 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 Create Symbol 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	Check to hide description by default. Once a symbol is dropped on a drawing, users can select to show or hide the description text by right clicking on the symbol and select "Edit Reference"
Insert Horizontal / Vertical rotation menu	To insert a horizontal / vertical rotation menu into 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.
Category	If you assign your symbol to a category, during component selection you will be presented with components for that category only. This makes for easier and faster component selection. Categories are guides only and during component selection the user can still select a component in other categories no matter what category a symbol belongs to.



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 one or more component during panel layout and reports.

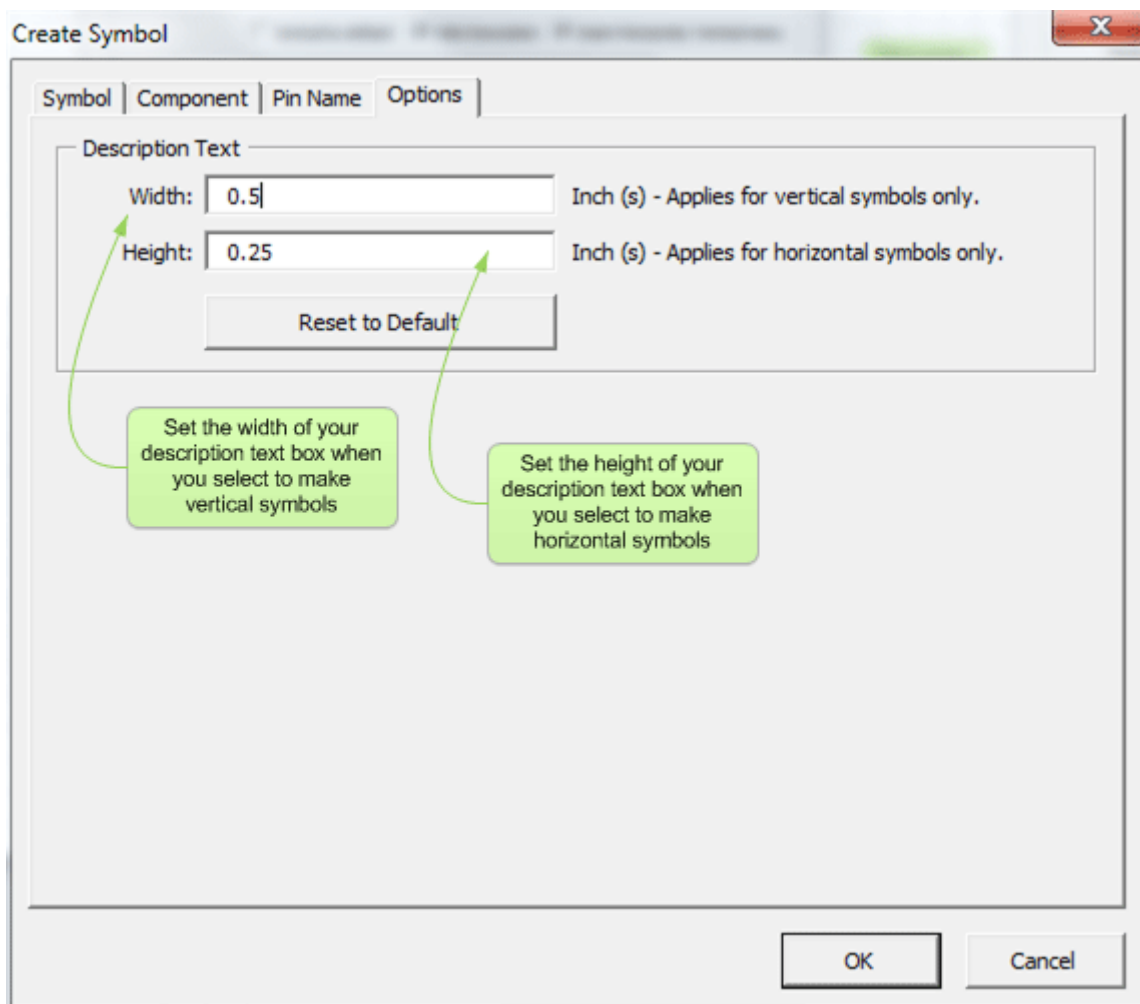
Priority	All schematic symbols in Electra have a priority number that is used to match a group of symbols with the same references to a component. The lowest priority number in a group of symbols with the same references will be used to suggest a default component. For user created custom symbols, a priority number of 15 is recommended.
Location Type	The Location Type combo box is used by the Location or AutoLocation 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.

See Also: Understanding Symbols and Components, How Symbols are Matched to Components

All symbols with connection points can have a set of pin names assigned to the connection points.

Pin Set	Each connection point has a pin name. If none were entered, Electra will use defaults (blank). Pin Names comes in sets. For example, a standard NO (2 pole) can have 13 / 14, 23 / 24 etc as pin names. 13 / 14 is one set of pin names and 23 / 24 is another set of pin names. To specify numerous sets of pin name for a single pin, type in all the possible pin names for that pin separated by commas, e.g. 13, 23. Then cycle to the next pin to repeat the same and type 14, 24 as pin names.
Position	Click on any 8 of the radio buttons to set the pin name position relative to the connection point. You will be able to see how your selection affects the position of the pin name on the right side of the above window.
Text Orientation	Select horizontal or vertical text for your pin name text orientation.

See Also: Displaying Pin Names, Automatic Pin Names, Assigning Pin Names



When you wanted detailed control the width and height of the description textbox, use the width and height text box.

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

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

To modify and edit your symbol:

- Drag your symbol onto your drawing
- Modify it and use the Create Symbol window again to add it back into a stencil

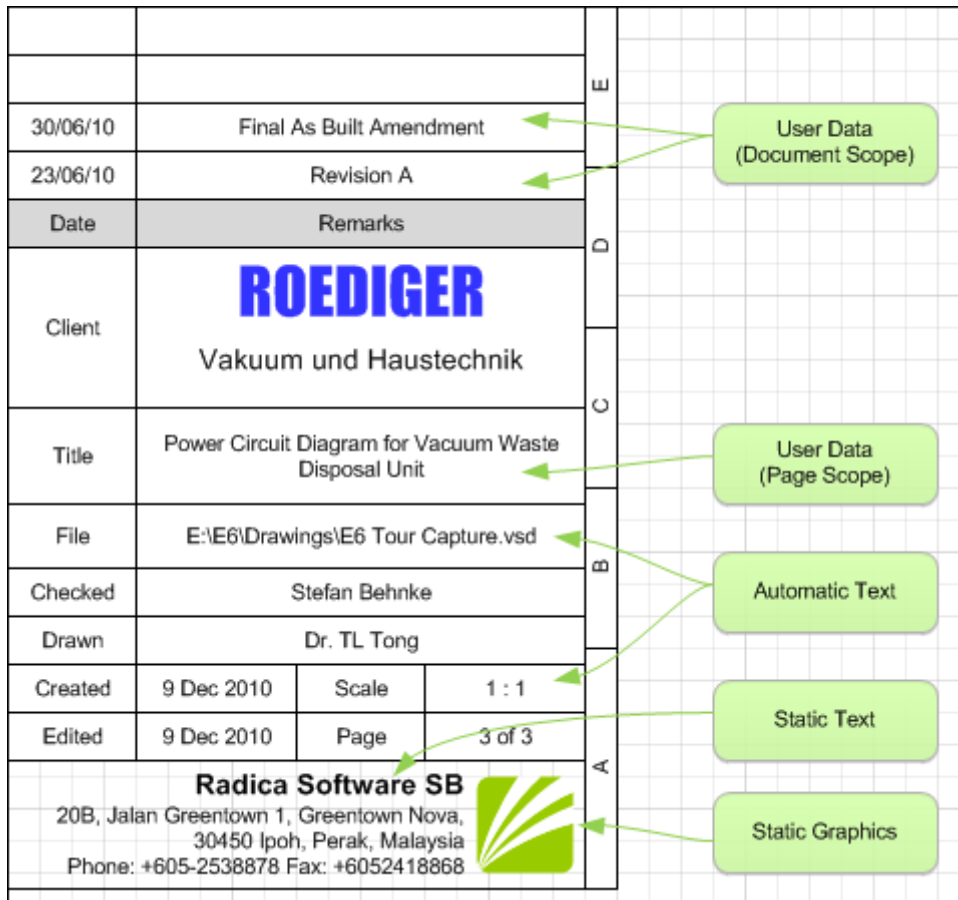
12.4 Editing the Symbol Icon

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 (In Visio 2010 it's called a preview).

13. Creating Your Own Title Blocks

13.1 The Anatomy Of A Title Block

A title block can consist of the following:



Static Text	Text that does not change over projects or pages, e.g. addresses or company names.
Static Graphics	Graphics that does not change. e.g. logos or client logos.
Automatic Text	Automatic fields that is automatically updated by Visio, e.g. editing dates, scale and page numbers.
User Data (Document Scope)	User editable data that applies for a single project, e.g. Project name, site name or any data that applies over the entire project.
User Data (Page Scope)	User editable data that applies for the current page only, e.g. page title, page author, etc.

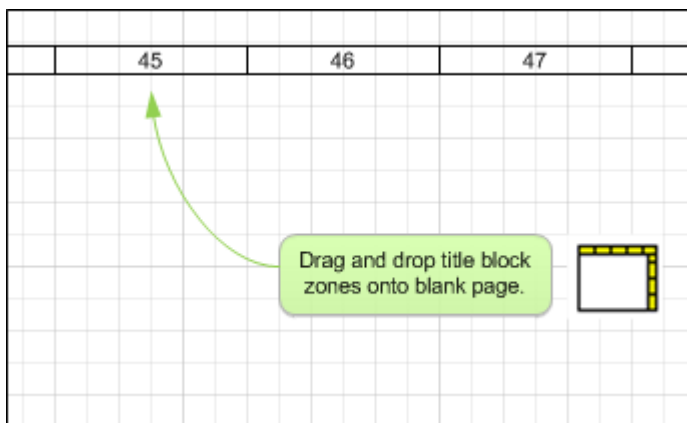
In Electra, there are tools that makes it easy to make your own custom title blocks that includes all of the above.

13.2 Title Block Creation Process

To create your own title block:

- Insert title block zones onto a blank page
- Insert static text and static graphics
- Insert automatic text
- Insert user data (page or document scope)
- Group all shapes together
- Select all and click menu Electra | Title Blocks | Create Title Block
- Type in a name and a prompt for your title block and click OK

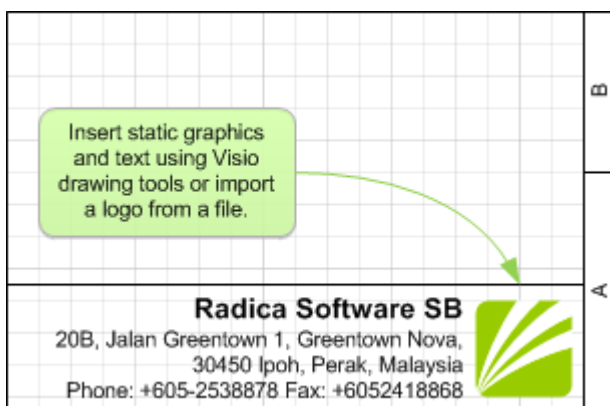
13.3 Inserting Title Block Zones



The title block zones automatically setup and divide your page into location zones based on your settings in menu Electra | Document Options.

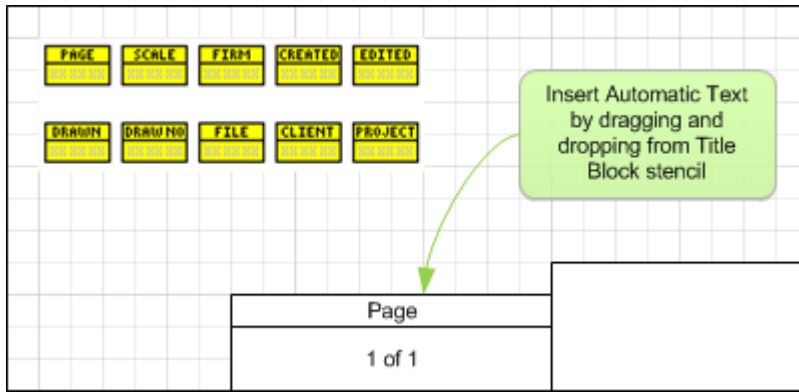
See Also: Location Zone Options

13.4 Inserting Static Text and Graphics

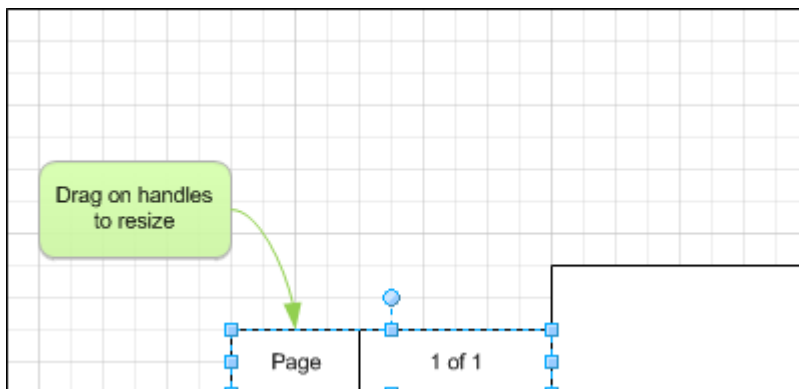
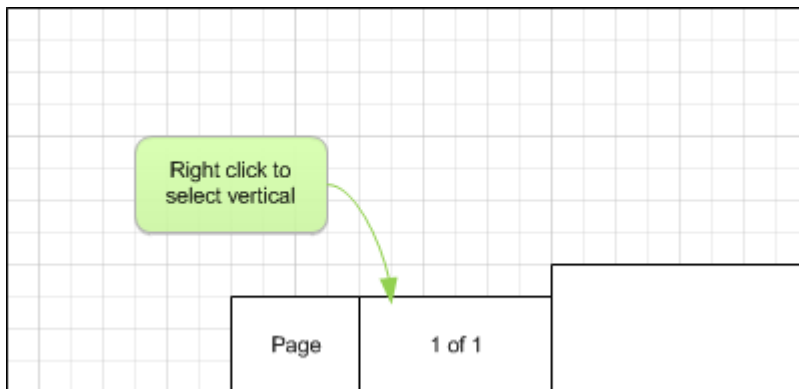


Use drawing and text tools on Visio to create graphics and text or import an image. If necessary, group them and place them in the desired location on your title block.

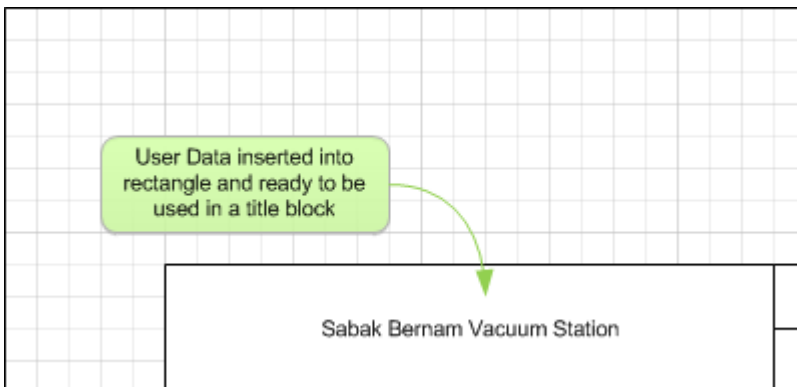
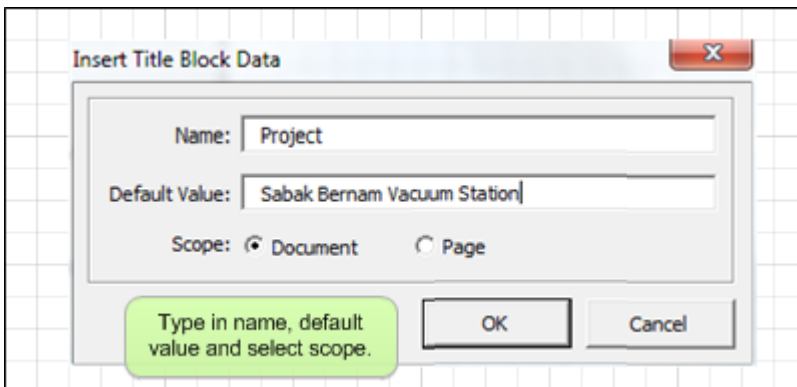
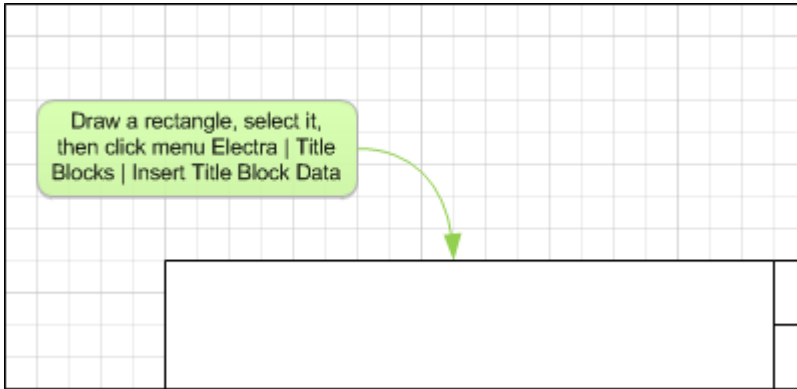
13.5 Inserting Automatic Text



Electra provides a host of automatic shapes including pages, scales and editing dates that can be placed on your title blocks. Just drag and drop them onto your drawing from the Title Block stencil. Right click on them for more options and place them in your desired location.



13.6 Creating and Inserting User Data

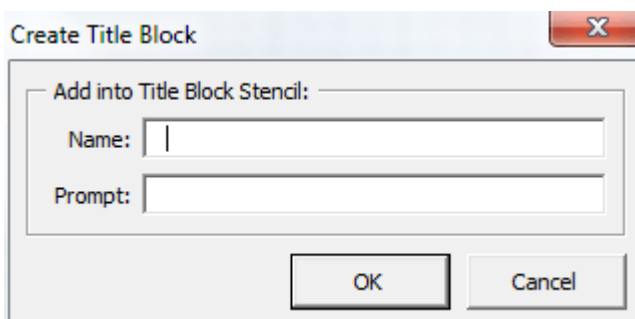


To create User Data:

- Draw a box and select it
- Click menu Electra | Title Blocks | Insert Title Block Data
- Type in a name for your User Data, e.g. Project or Page Author
- Type in the default value for the User Data
- Select Document scope for data that applies throughout the document or Page scope for data that applies for current page only

The User Data rectangle is now ready to be used on your custom title block. Drag the handles of the rectangle to position and resize your user data. Upon completion, users can edit these User Data by right clicking on the title block.

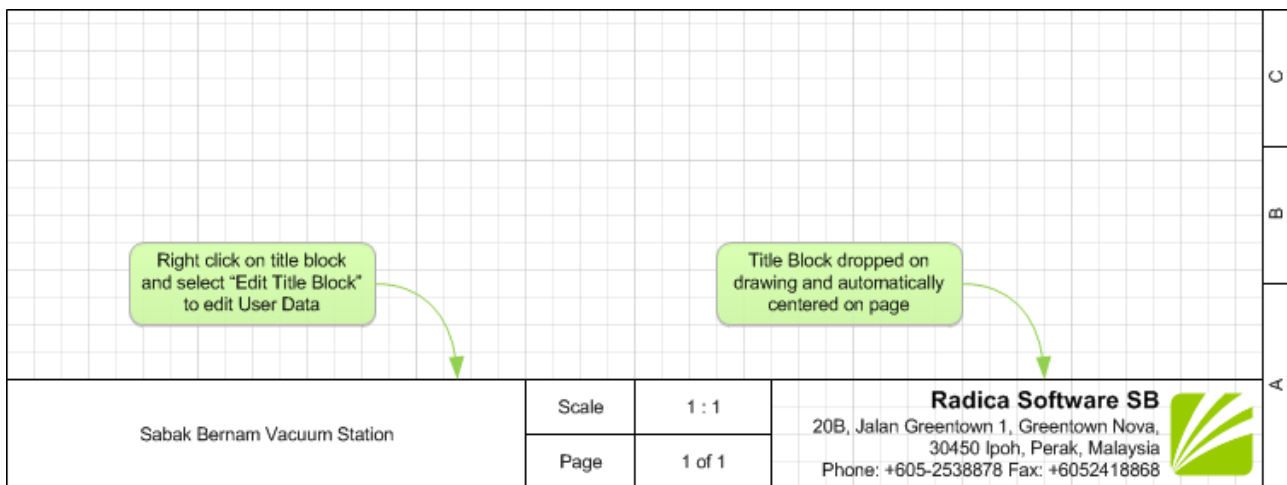
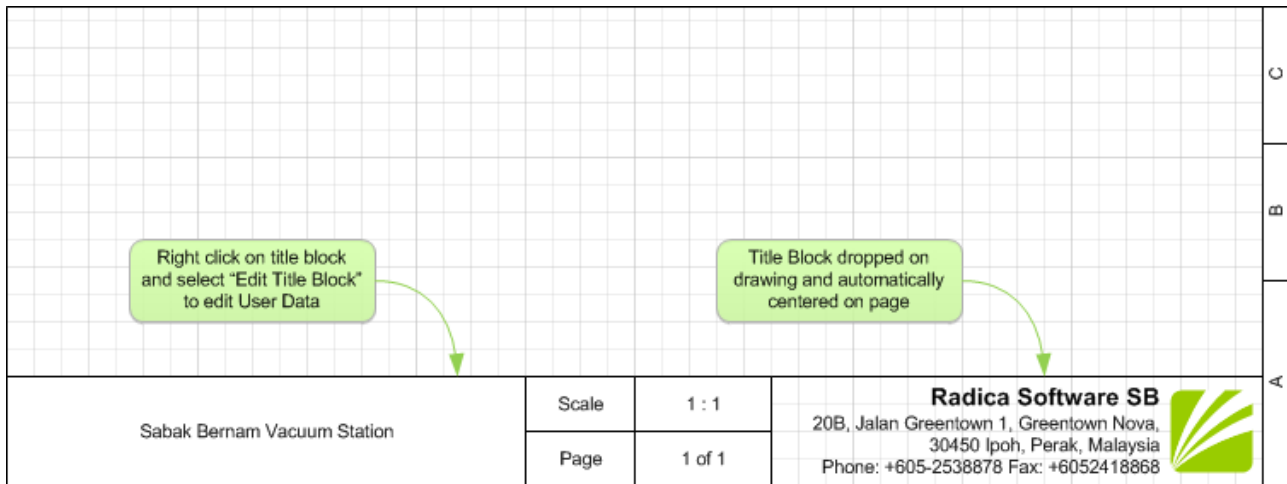
13.7 The Create Title Block Window



After selecting all shapes that make up your custom title block, group them together, then click on menu Electra | Title Blocks | Create Title Block. Type in a name and prompt for your custom title block, and click ok.

Your new custom title block will now be saved and stored in the Title Block stencil ready to be used.

13.8 Using a Custom Title Block



To use your custom title block:

- Drag and drop your custom title block from stencil onto your drawing

Your custom title block will automatically center on your drawing.

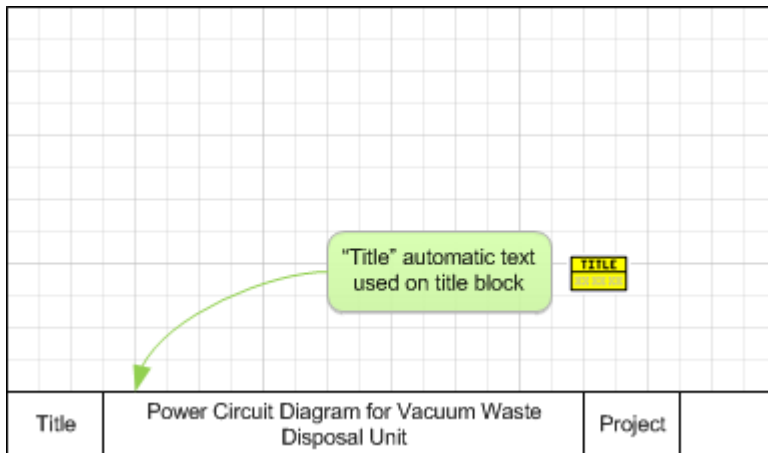
To edit and modify User Data:

- Right click on the title block and select "Edit Title Block"

13.9 Editing the Title Block Icon

An icon will be generated automatically when the title block is added to a stencil. Right click on the title block on the stencil and select Edit Icon to modify the icon. After creating your custom icon for your title block, right click on the title block in the stencil, select Master Properties, and make sure to deselect Generate Icon automatically from shape data (In Visio 2010 it's called a preview).

13.10 Generating Table of Contents



To use Table of Contents:

- Users must use the "Title" Automatic Text block from the Title Block stencil
- Insert the "Title" Automatic text into your Title Block and create your own title block
- Insert your custom title block into drawing
- Insert Table of Contents (TOC) report from the Reports stencil into your drawing
- Right click and select "Generate Report" and Electra will automatically scan every page to create a Table of Contents

The diagram shows a grid background with a callout box containing the text: Table of contents (TOC) shape dropped on drawing. Right click to generate. Below the grid is a table representing a Table of Contents report:

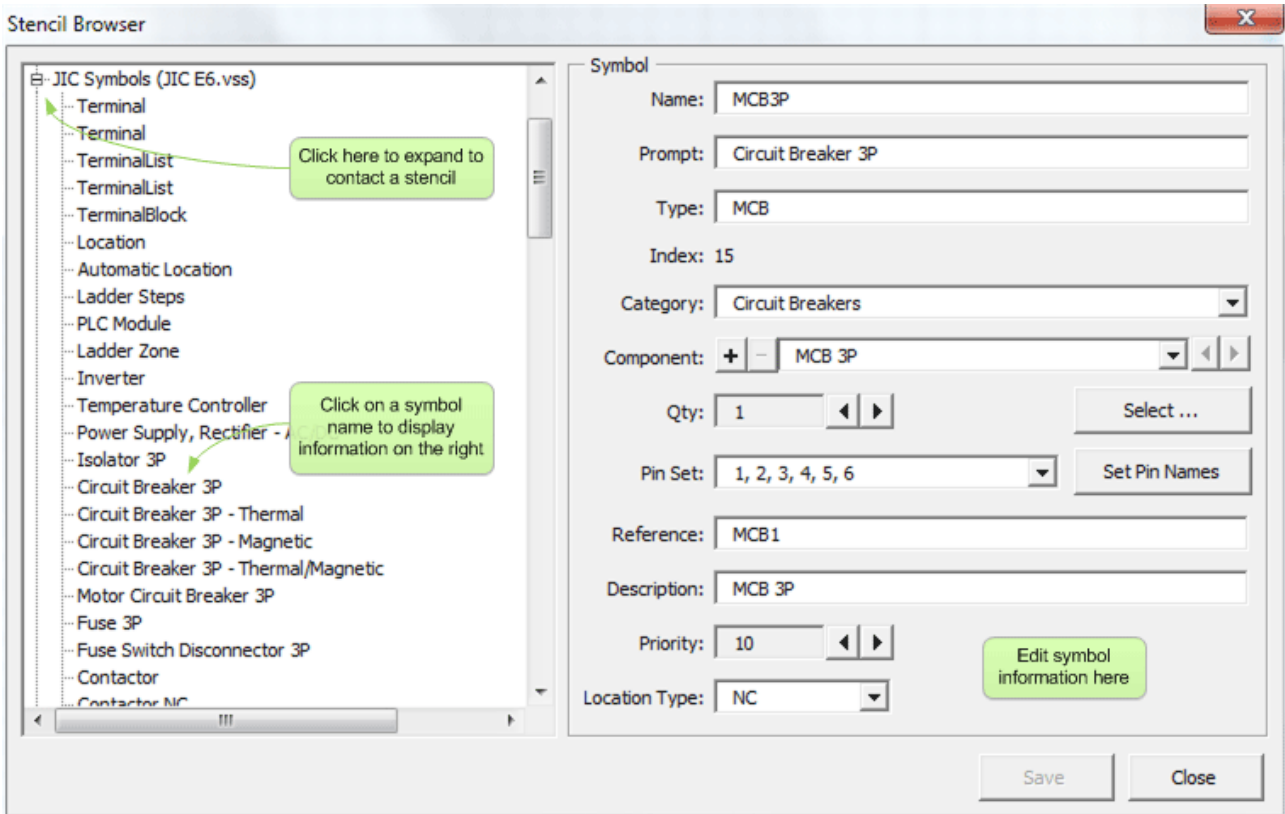
Table of Contents	
Item	Title
1	Power Circuit Diagram for Vacuum Waste Disposal Unit
2	Power Circuit 1
3	Power Circuit 2
4	Fan Control Circuits
5	PLC Circuits
6	Terminal Layout
7	Layout
8	Front Panel Layout
9	Cross Reference Report
10	Material Reference Report
11	Bills Of Materials Report

14. Miscellaneous

14.1 Stencil Browser

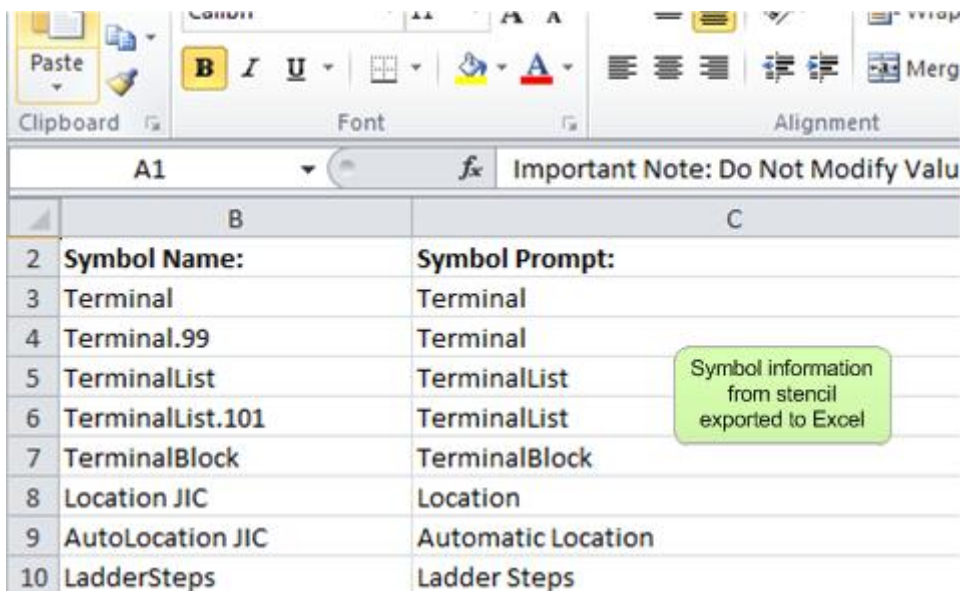
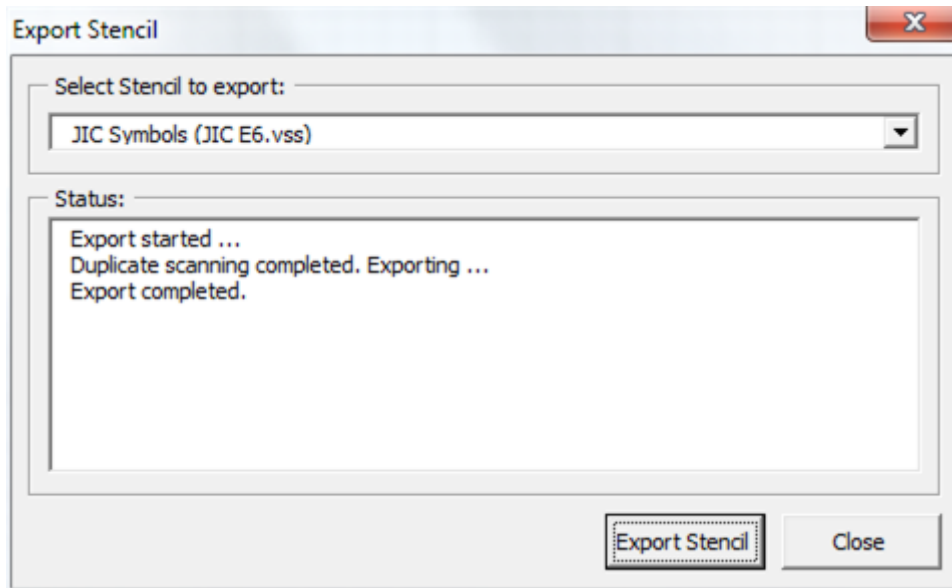
To access the stencil browser:

- Click on menu Electra | Stencil Browser



The stencil browser is a utility to edit symbols after their creation. Users could browse stencils containing symbols and edit their component or pin information without recreating a symbol.

14.2 Exporting Stencils



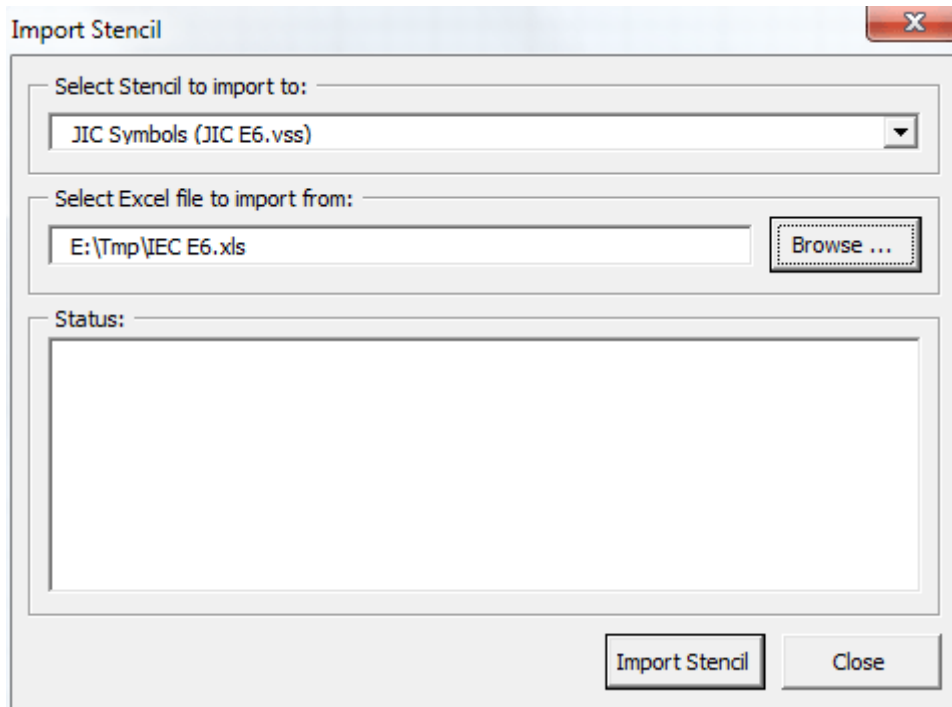
	B	C
2	Symbol Name:	Symbol Prompt:
3	Terminal	Terminal
4	Terminal.99	Terminal
5	TerminalList	TerminalList
6	TerminalList.101	TerminalList
7	TerminalBlock	TerminalBlock
8	Location JIC	Location
9	AutoLocation JIC	Automatic Location
10	LadderSteps	Ladder Steps

To export a stencil:

- Click on menu Electra | Export | Stencils
- Select the stencil to export

Exporting stencil and symbol information to Excel enable users to edit these information in batches rather than individually using the Stencil Browser.

14.3 Importing Stencils

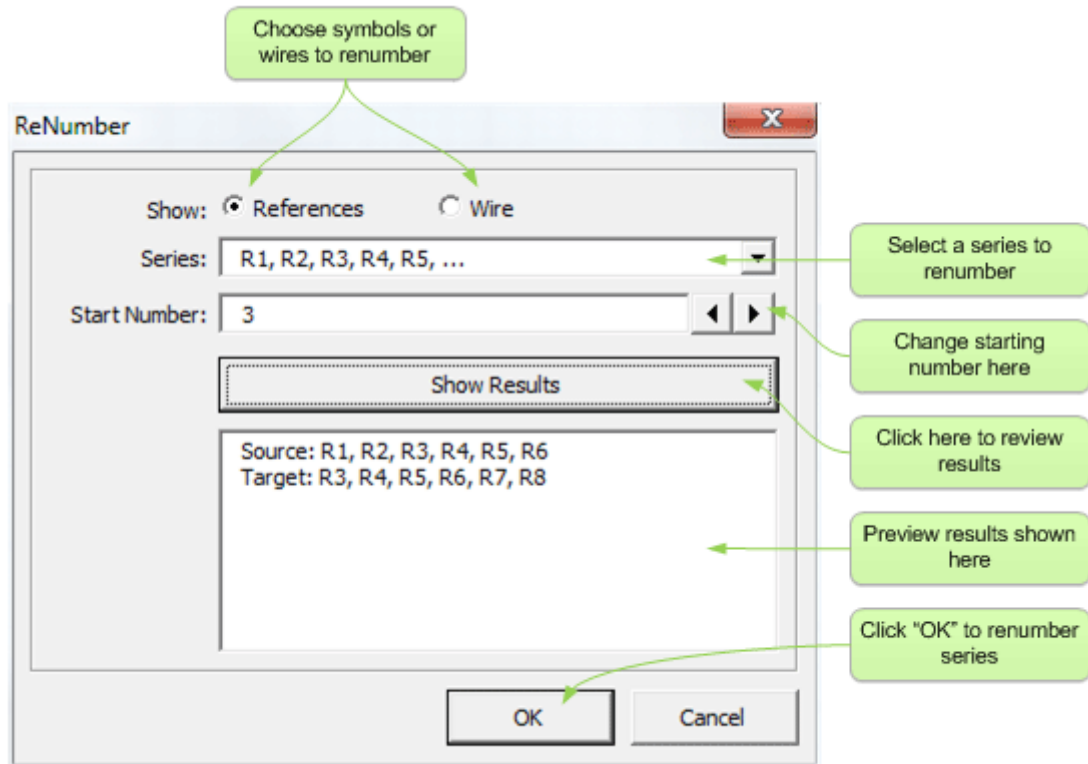


After exporting symbol information from a stencil into Excel, and upon completion of editing these information, they can then be imported back into Electra:

- Click menu Electra | Import | Stencils
- Select to stencil to import into
- Select the Excel file that contains symbol and stencil information
- Click OK

Updated symbol information will then be automatically imported to the stencil.

14.4 The Renumber Utility



The Renumber utility is used to renumber a batch of symbols or wires with the same format into other numbers.

14.5 Clean Up Document

When a user drops a Master shape or symbol from a stencil onto a drawing, Visio automatically inserts the master shape into a special stencil called the document stencil. When a user deletes a symbol from a drawing, these Master shapes get left behind in the document stencil even though they are no longer used and causes the entire document to increase in size.

Run the "Clean Up Document" utility, and Electra will scan the entire document and remove all unused Master shapes to reduce file sizes.

15. Appendix A: Installation and Activation

15.1 System Requirements

Electra requires:

- Microsoft Visio 2003 or later
- Works with all versions of Visio including standard, professional, technical and academic editions (only 32-bit versions)
- DOES NOT WORK on all Office / Visio 64-bit versions as Microsoft has not worked out compatibility problems with 64-bit versions. Microsoft has recommended AGAINST installing Office 64-bit unless absolutely necessary. Please refer to the below for more information:
- [Microsoft TechNet](#)
- [Microsoft TechNet Blog](#)
- [PC World](#)
- Works with Visio (32-bit) on all platforms including Win XP, Win Vista (x86 & x64), Win 7 (x86 & x64)

15.2 Installing Electra

Electra is a Microsoft Visio solution and requires Microsoft Visio to operate. It is strongly recommended that you install Visio before installing Electra.

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

To install Electra:

- Exit any applications you are running
- Double click on the installer and run the installer program
- Read the installer warning regarding closing any applications that you may be running and click OK
- Read the End User License Agreement and if you agree to the terms, click Accept
- Click on the installation button to begin installing

15.3 Activating Electra

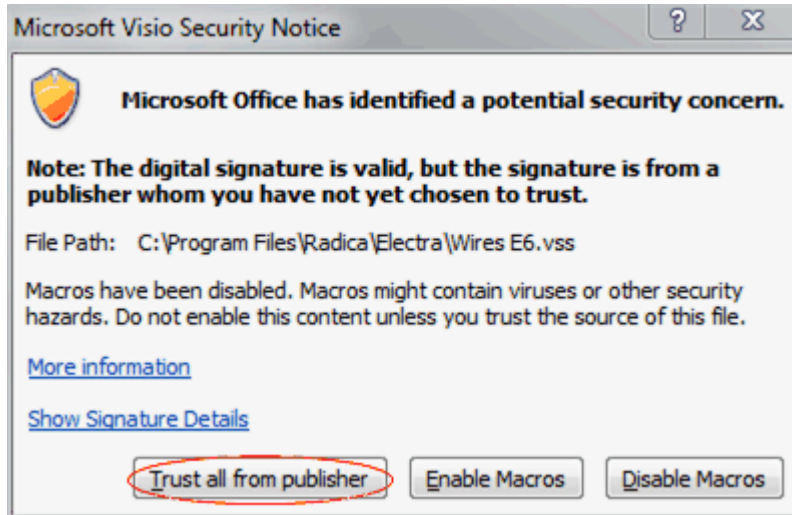
To activate your copy of Electra:

- Click menu Electra | About Electra
- Click on Edit button and paste/type in your serial number, then click Save
- Click on Activate button to activate

15.4 Enabling Macros

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:

- Start Electra by double clicking on New Electra Drawing.vst
- Visio will ask you about trusting us as a publisher, as listed below. Click “Trust all from publisher”



- In Visio 2010, you will also be asked to turn on macros as listed below. Click on “Enable Macros”

