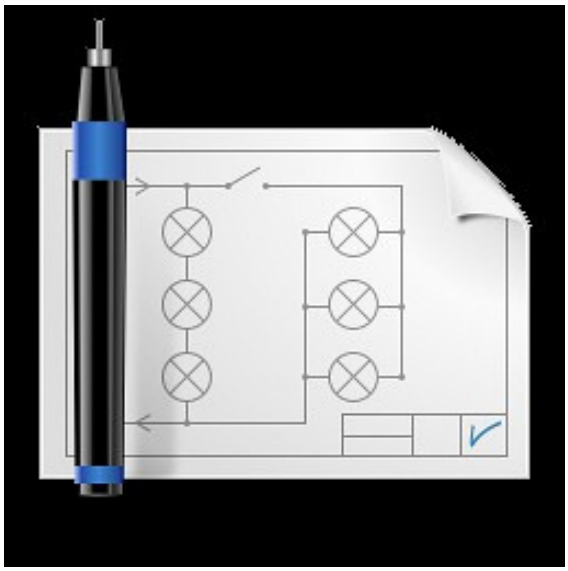
QElectroTech - a quick start guide [1]



QElectroTech
Version 0.4-dev

February 27, 2014

[1] Document and graphics typeset in LaTeX in GNU Emacs-Auctex; created under Creative Commons Attribution 3.0 License

Author

The team QElectroTech

License

This work is licensed under the Creative Commons Attribution 3.0 base License

http://creativecommons.org/licenses/by/3.0/deed.en

Logo

Nuno Pinheiro ✉ **nuno@nuno-icons.com**

QElectroTech

Home Project: www.qelectrotech.org

Authors QElectroTech

Original idea: Benoit Ansieau ✉ **benoit@qelectrotech.org**

Collection of elements and Development: Laurent Trinques ✉ **scorpio@qelectrotech.org**

Development: Cyril Frausty ✉ **cyril@qelectrotech.org**

Development: Joshua Claveau ✉ **Joshua@qelectrotech.org**

Development: Abhishek Bansal ✉ **abhishek@qelectrotech.org**

Documentation

Documentation English: Eswara Arun Kishore ✉ **arunkishoree@imu.co.in**

Contents

1 Introduction . . . 1
1.1 About QElectroTech . . . 1
1.2 Installation . . . 1
1.3 Comments . . . 1
1.4 Creating diagrams . . . 1
2 Description of the drawing window . . . 2
2.1 Title bar: . . . 2
2.2 Main Menu bar: . . . 2
2.3 Tool bar: . . . 2
2.4 Panel of elements: . . . 3
2.5 Control tabs: . . . 4
2.6 Help bar: . . . 4
2.7 Project title tab: . . . 4
2.8 Scheme title tab: . . . 5
2.9 Workspace: . . . 5
3 Creating new project . . . 6
4 Creating a new diagram . . . 7
5 Configuring QElectroTech . . . 8
6 Inserting elements . . . 10
6.1 Folio report . . . 10
7 Text field . . . 13
7.1 Inserting text field . . . 13
7.2 Orientation of the text field in work space . . . 13
7.3 Moving and editing text field in work space . . . 13
8 Working with connectors . . . 14
8.1 Re-sizing connectors . . . 14
9 Element's editor . . . 16
9.1 Creating a new element . . . 16
9.2 Description of Element Editor . . . 17
10 Sample tutorial - Creating Globe valve element . . . 23
11 Title block template editor . . . 29
11.1 Opening title block editor . . . 29
11.2 Title block editor features . . . 30
11.3 Custom tab . . . 32
11.4 Working with the title block editor - (a tutorial) . . . 33
12 Printing, Exporting and Saving . . . 38
12.1 Printing . . . 38
12.2 Saving . . . 38
12.3 Exporting . . . 38
13 Some sample drawings created in QElectroTech . . . 40

1 Introduction

1.1 About QElectroTech

QElectroTech is an application to create electrical drawings. Some functional mechanical objects are also provided. QElectroTech has an element editor, which permits creation of elements like switches, valves etc., as per user's requirements. This feature enables QElectroTech to facilitate 2D sketching of electrical, hydraulic, mechanical and pneumatic systems. It is a good option for P&ID drafting and can render output to wide variety of formats like 'dxf', 'pdf', 'jpg', 'png' etc.,. A number of international languages are also supported in QElectroTech for preparing electronic / electrical circuit diagrams. Elements created in QElectoTech are saved in XML format. QElectroTech is available for both MS Windows and Linux operating systems.

QElectroTech is Free Software released under the GNU / GPL license. As on the date, a stable version 0.3 is released for systems operating on MS Windows, GNU / Linux and MacOS. The present English documentation is developed for 0.4-dev version, which is yet to be released. However, this document can serve as a complete documentation for version 0.3 as well.

1.2 Installation

You can download the latest version of QElectroTech from http://qelectrotech.org/download.html. For GNU/Linux systems source files can be downloaded and configured. Ready made packages for some distros are also available for download.

For MS Windows systems, ready to use executable installation files (.exe) are available for download which are packed in 'zip' folder.

1.3 Comments

A collection of articles with the current version of QElectroTech are given without any warranty. It allows you to edit, modify and use the items without conditions and regardless of the final license.

1.4 Creating diagrams

In the case of using all or part of the QElectroTech library for purposes other than to create wiring diagrams, you must abide by the terms of the Creative Commons-by license: This work is licensed under the terms of the Creative License Commons Attribution 3.0. For a copy of the license please visit the website: http://creativecommons.org/licenses/by/3.0/ or send a letter to Creative Commons, 171 Second Street, Suite 300, San Francisco, California, 94105, USA.

2 Description of the drawing window

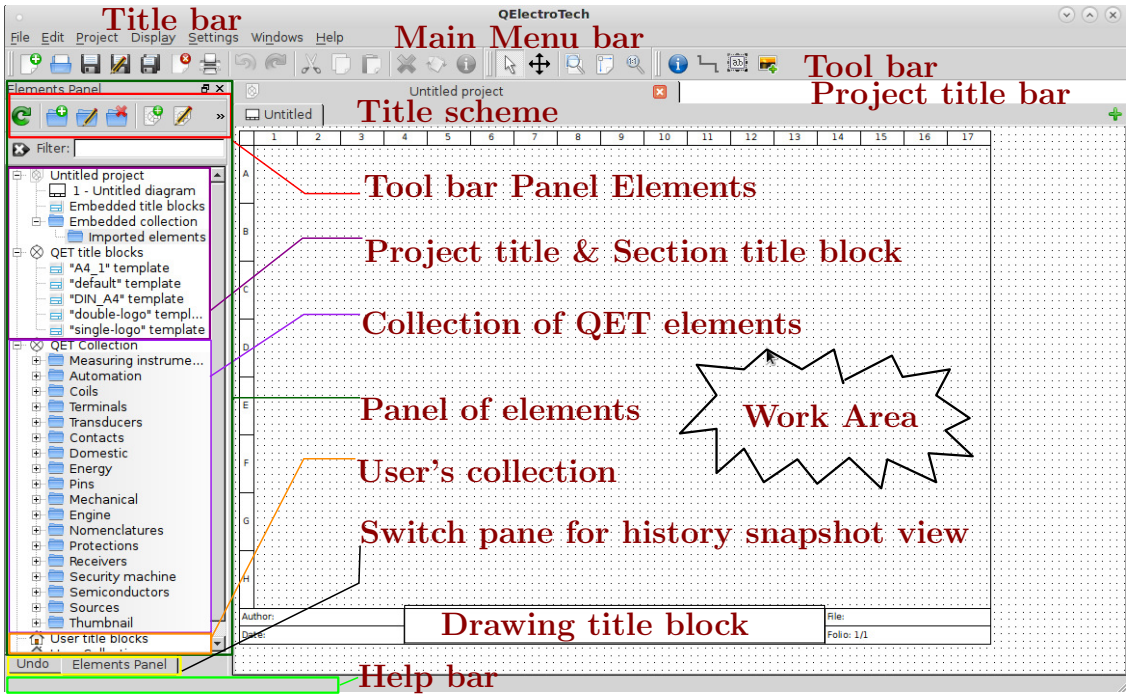


Figure 1: QElectroTech editor main window

2.1 Title bar:

It contains the standard controls for QElectroTech, like 'maximize', 'minimize', 'resize', 'move' or 'close' among others. The controls appear by 'left clicking' the left corner of the title bar.

2.2 Main Menu bar:

It is a standard menu bar, which allows you to access all the features of the application such as 'File', 'Edit', 'Project', 'Display', 'Settings', 'Windows' and 'Help'.

2.3 Tool bar:

It contains buttons for basic functions in the drawing window. Three separate bars make up the 'Tool bar' as shown in Figure 1 . All bars begins with a '‖' marker. The tool bars can be moved by left clicking these markers & drag dropping to a new location. Each bar is further segregated by separators to group shortcuts according to the field of operation. Options like 'new' diagram etc., that effects the entire window are grouped together separated from options like 'cut' etc., which operate only on objects in diagram. The tools available are discussed below with graphics.

Figure 2: Tool bar

The first tool bar is shown in Figure 2 and 3. It contains the standard tools available with regular programs. The options include shortcuts for starting a 'new diagram', 'opening a saved diagram', 'saving changes to the active diagram', 'closing the drawing', 'printing the present diagram', 'undo' and 'redo' changes, 'cut', 'copy' and 'paste'. These options are available in the left to right order on the tool bar. The icons that operate on objects in the diagram are not active in the Figure 2. The 'undo' shortcut activates with the first input to the drawing and 'redo' is activated only after an 'undo' action is performed. Other shortcuts become active only when object(s) is/are selected in the drawing.

Figure 3: Tool bar

The last segment of the first bar and the remaining two bars, which constitute 'Tool bar' are shown in Figure 3. Shortcuts for exclusive editing and building your diagram are provided here. 'Delete', 'rotate element(s)' and 'selection properties' are available on the first segment. An element properties can be viewed by selecting it and left clicking the 'selection properties' button. Element editor can be launched from the properties window by clicking the 'Edit element' button on the properties window. The elements from the QET collections are read only and cannot be edited. However, an element can be imported into 'User's collection' and edited. Please refer section 9.1 on page 16, for guidance on editing or creating elements.

The second bar contains buttons to 'select object(s)', 'Move', 'zoom to contents', 'fit in view' and 'default zoom'. Select object(s) allows selection of one or more objects at a time. To select more than one element, hold 'control' key from keyboard and left click the elements. 'Move' option facilitates to view the entire drawing without modifying it, it shuffles the drawing area on the screen. In physical terms it can be described as moving the entire diagram sheet to view contents. 'Zoom to contents' zooms view to the extent of all the elements placed in the diagram. 'Fit in view' sizes the drawing to display complete diagram in the window. The 'default zoom' option sets the zoom level of the diagram to a default value.

The third bar has options to set 'diagram properties', 'reset conductors' connecting the objects, 'add a text field' to the diagram and to 'insert picture' in the diagram. 'Reset conductor' option is activated only when one or more conductors are selected. The description of each of these functions is explained in the later sections of this document.

2.4 Panel of elements:

The dark green colored box (refer Figure 1) in the panel contains essential tools for drawing in the 'Work Area'. The panel consists of the following categories of elements:

Tool bar panel elements:

The 'tool bar panel elements' is organized into three groups separated by markers. The first group has a shortcut to reload all collections; second group has options to create new category, edit category and delete category; third group has options to create a new element, edit an element and a button to open a new element file.

Filter:

A filter is provided to quickly search elements by searching for the input keywords is provided. When a keyword is entered, each matching element is displayed along with its tree, i.e under the category in which the element is located. Only the categories which hold a matching element for the searched keyword are displayed in the project title section and in the collection of QET elements (refer Figure 1). The entered keyword can be cleared by 'left clicking the '*X*' button (provided left to the search keyword text input field), to display normal categories and QET elements. The keyword can also be cleared by using backspace key from keyboard or selecting the text with mouse and deleting the entered keyword(s).

Document and graphics created under Creative Commons Attribution 3.0 License

Project title & section title block:

Project title can be given the drawing title name in the 'diagram properties' or by double clicking the 'drawing title block'. There are other options that can be set in this action. Once a name is given to the drawing, it gets displayed in the 'project title' block. The project title section gets updated as the drawing is made. The 'title block' has templates on which the drawing is made. Standard templates are provided with installation with the template set to 'default'. Other templates can be applied to the drawing by selecting the template and drag dropping it on to the 'work area'. The templates can be edited from 'diagram properties' using the drop down clickable button provided right to the template field.

QET collection:

QElectroTech provides a set of readily available elements grouped under a relevant category, to quickly place elements in the drawing. Elements can be selected and drag dropped onto the work area to prepare the diagram. An element can be quickly searched using keywords in the filter.

User's title block:

User can add a title block (template) by 'right clicking' user title block and selecting 'new template' and choosing a new name. The title block can then be set according to the users choice for continued use in future projects. User has many options like adding a logo and details to the template alongwith many other editable options.

User's collection:

'Right clicking' the user collection gives options to the user to add new categories, elements and to delete categories. User can import elements from QET collection and customize the elements for use in a project. The user also has option to import new elements stored in a file. This is category under which user can create customized categories and elements.

2.5 Control tabs:

To the bottom of the elements panel, two tabs 'undo' and 'elements panel' are provided. Elements panel is default view in QElectroTech. 'Undo' tab records each change to a diagram in the chronological order. At any point if the user wishes to go back to any previous point in the diagram, he can select the tab 'undo' select a point in the history where he/she wishes to return and switch back the tab to 'elements panel' and continue drawing. The changes that were effected after the event will be undone. ⚠The changes will be lost with this action. After the actions are undone, user will find that the actions that were undone are highlighted in light pink color, under appropriate category in the elements panel. These actions are marked internally by QElectroTech as unused in the project. The tabs can be turned off or on from the main menu bar Settings —> Display. Unchecking a choice would turn off the tab in the window.

2.6 Help bar:

The help bar is the space below the control tabs. It is very useful for beginners of QElectroTech in the way that it gives information about the field that is pointed by the cursor. A user can learn about a field by simply pointing it with the mouse and looking at the help bar.

2.7 Project title tab:

By default, QElectroTech displays the project name as 'untitled project' in the project title tab. The user can save the project by giving it a name of his/her choice. The project title tab displays the name of the project, which is saved as *.qet file on the computer. There is an option to close the project, by clicking at the top right corner of the project title bar. A new project can be opened in the same window, which opens as a new tab in the project title. User can switch from one project to the other that are opened in the same window by left clicking the tab with the mouse or pressing 'control + tab' key from keyboard. The project title can be changed from main menu Project —> Project Properties and entering a name for the project in the project title field.

Document and graphics created under Creative Commons Attribution 3.0 License

2.8 Scheme title tab:

By default the scheme title tab shows 'Untitled'. Double left clicking the tab opens the diagram properties window, which has a field to assign a name to the drawing among other options to set author's name, date etc.,. The tab shows the name of the drawing. A new drawing to the present project can be added by right clicking the project in the project title and section title block and selecting 'Add a diagram' option in the popped up window. The same action can also be achieved by clicking the ✚ provided at the right end of scheme title tab. Each scheme tab is one drawing in a project and many drawings can be added to a project. Also, each drawing can be assigned its own properties from the diagram properties window.

2.9 Workspace:

By default, work space defined by 'A' to 'H' rows and '1' to '17' columns. The rows and columns can be added or reduced from the diagram properties window. The default size of each column is 60 pixels and that of each row is 80 pixels, both of them can be adjusted in the diagram properties. Workspace is the area where the diagram is prepared, by including elements, conductors, text and pictures.

Document and graphics created under Creative Commons Attribution 3.0 License

3 Creating new project

A new project can be opened from File —> New or by clicking the shortcut icon
Next click Main Menu —> Project —> Project Properties; a pop up window for project properties opens up, which is shown below. A name for the project created can be given in the 'Project Title' field and click 'OK' to continue

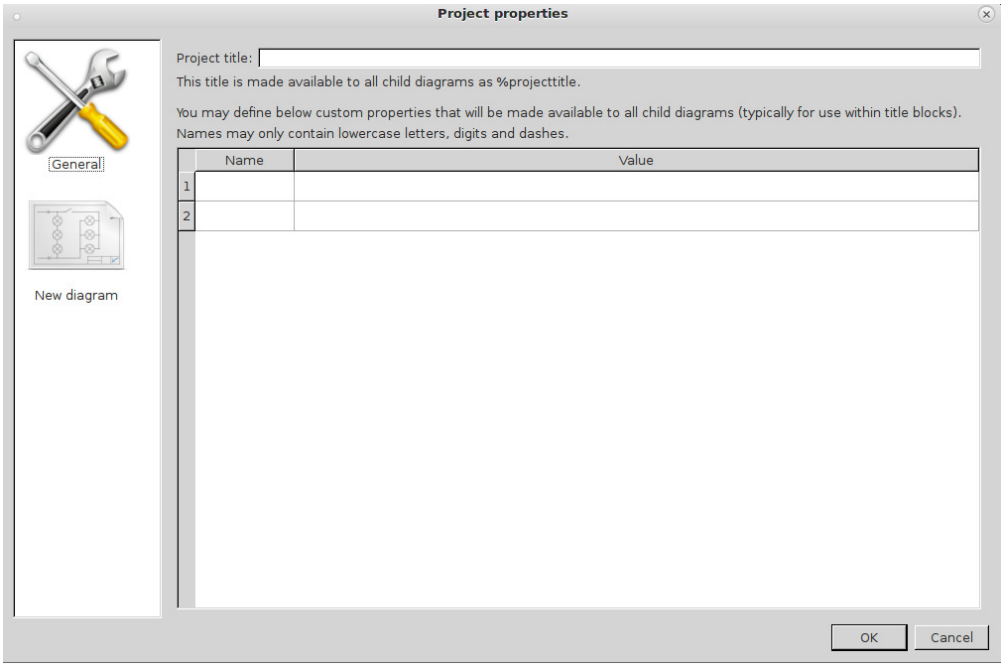


Figure 4: Project properties window

drawing. The project title tab now displays the project name and comments '[modified]'. The project should now be saved as a file 'file_name.qet', the project title tab then shows Project "project_name". In the above window if the properties for 'New diagram' are set, they are applied only to subsequent new schemes added to the project. The present scheme properties should be set from the 'Diagram properties' option from the tool bar or by double left clicking the scheme title tab.

Document and graphics created under Creative Commons Attribution 3.0 License

4 Creating a new diagram

By default, QElectroTech opens a “Untitled” diagram. Diagram name and other properties can be set by invoking diagram properties window. Options to invoke the diagram properties include Double left clicking the scheme title tab, selecting the ‘Diagram properties’ from the tool bar, from main menu bar Edit —> Diagram properties or by using ‘Control + L’ keys from key board. The diagram properties window has options to assign a name to the diagram, author’s name, date, changing the dimensions of the work area by adjusting number and dimensions of rows and columns, or selecting a standard template for the drawing etc.,. There is a custom tab provided for versions from 0.3, to define and use user defined keys.

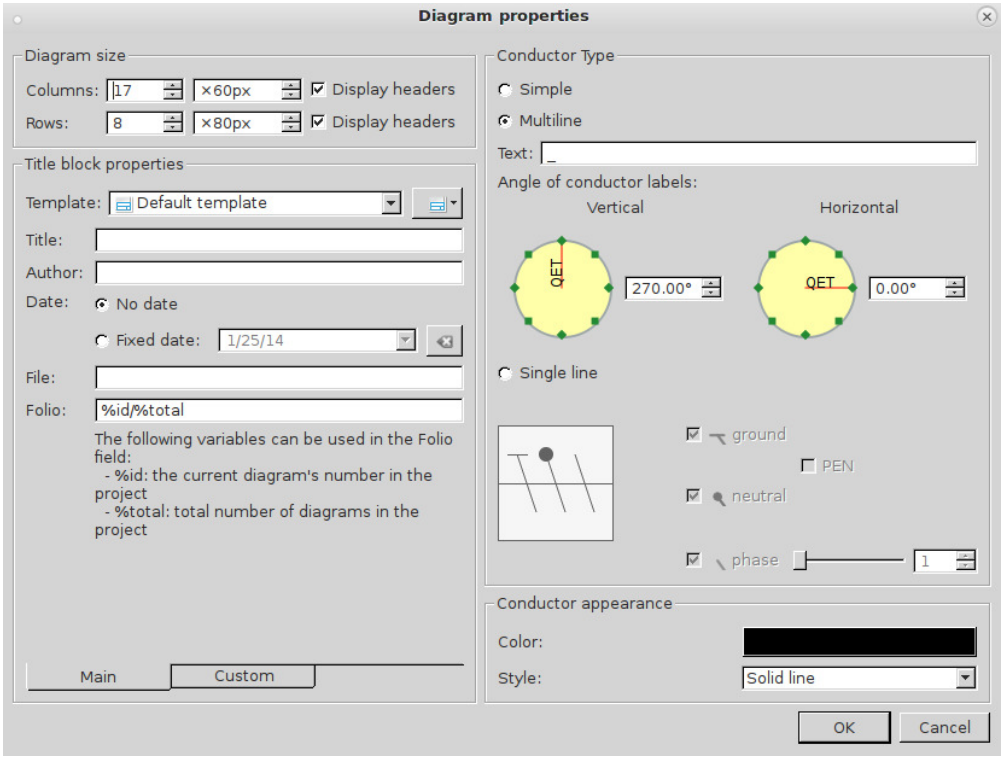
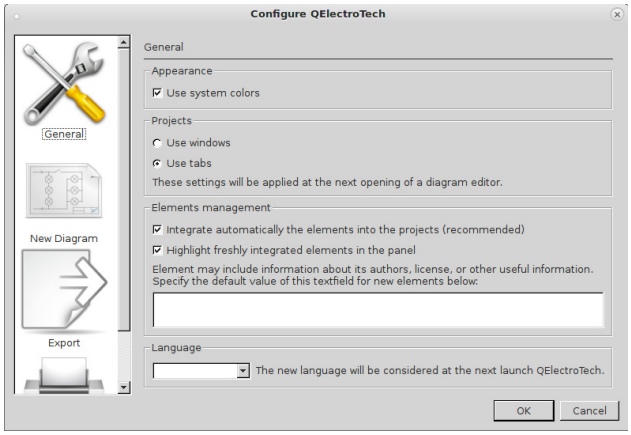


Figure 5: Diagram properties window

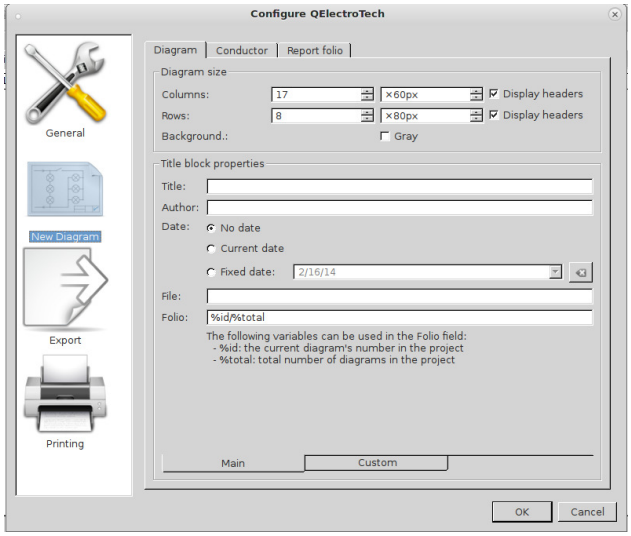
The diagram properties is explained further along with the use of custom tab in section 11.3 on page 32. The conductor type and properties options are explained under section 8.1 on page 15.

5 Configuring QElectroTech

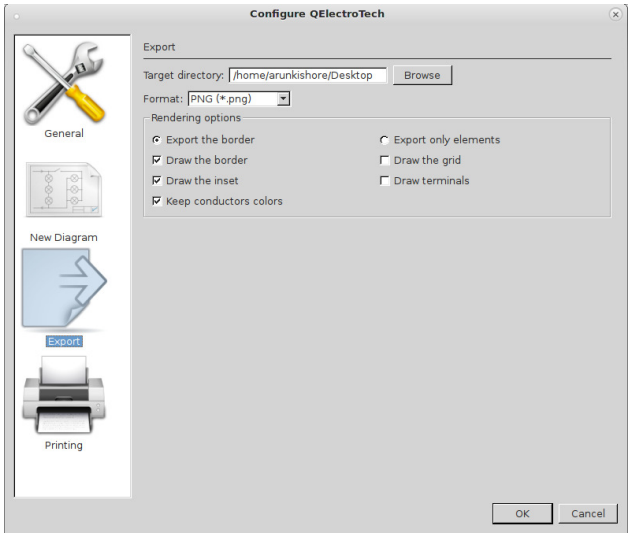
QElectroTech permits customizations through configure option. To access the configuration wizard, left click 'Settings' from main menu bar and select 'Configure QElectroTech'; (Settings —> Configure QElectroTech). The wizard permits setting users preferences for QElectroTech covering - 'General', 'New Diagram', 'Export' and 'Printing' aspects. The options are more or less self explanatory.



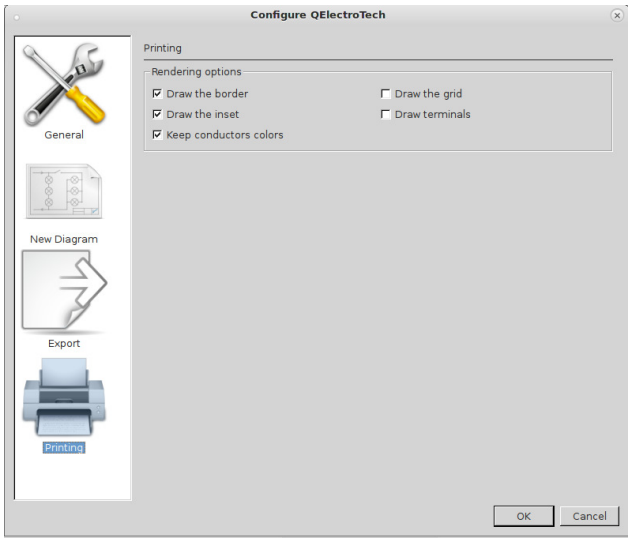
'General' options



'New Diagram' options



'Exporting' options



'Printing' options

1. **General options :**
   (a) **Appearance:** Check this option to use system colors for QElectroTech
   (b) **Projects:** Check this option how windows (different diagrams in projects) should appear in QElectroTech. Tab option is default and is illustrated in this documentation (Pictures of QElectroTech). 'Window option' causes diagrams to appear in windows and new additions show up in new windows.
   (c) **Elements management:** The options provided are to integrate elements and new elements created automatically to projects, default option to highlight newly added elements and to set default author and credits options for new elements created.

(d) **Language:** A language combo box lets to set default language for QElectroTech. For English choose English (Great Britain country flag).

2. **New Diagram:** Please refer section 4 on page 7. The New diagram is composed of three tabs the first two are the same as displayed for 'Diagram properties'. An additional tab 'Report folio' lets you use the defined names for values in report folio. An explanation to report folio is provided in section 6.1 on page 10.

3. **Exporting options:** QElectroTech default exporting properties for diagrams can be set here. Options include selecting an output format ('PNG' format is shown here), default location of exported files and rendering options. Rendering specifies the kind of information to be included in the output file.

4. **Printing options:** QElectroTech can print a drawing to a physical printer connected to the computer. The options that can be specified in the configuration window under print options are the rendering options. There are five options provided - each specifies whether to include the option in the print output. Checking an option, includes the data requested in the physical print diagram.

Document and graphics created under Creative Commons Attribution 3.0 License

6 Inserting elements

QElectroTech has a number of elements listed as a tree under QET Collection, under relevant categories in the collection of elements field shown in Figure 1. Alternatively a element can be searched quickly using 'Filter' field. Each element can be selected using left mouse click and drag dropped on to the work area to include it in a diagram. The elements can be positioned any where in the work area and rotated in quantum steps of 90°. Rotation of elements can be performed by selecting the element in the work area with left mouse click and pressing 'space' key from key board. Alternatively, rotate option in the tool bar gets active upon selecting at least one element in work area, which can be clicked to orient the selected element(s) to the required rotation. A number of elements can be selected together by holding 'control' key from key board and left clicking required number of elements in the work area. Once the required elements are selected (evident from light gray box enclosing each selected element), rotation operation can now be performed on all the selected elements together as described earlier. You will also find some elements like for example, a horizontal ammeter that cannot be rotated for obvious reasons.

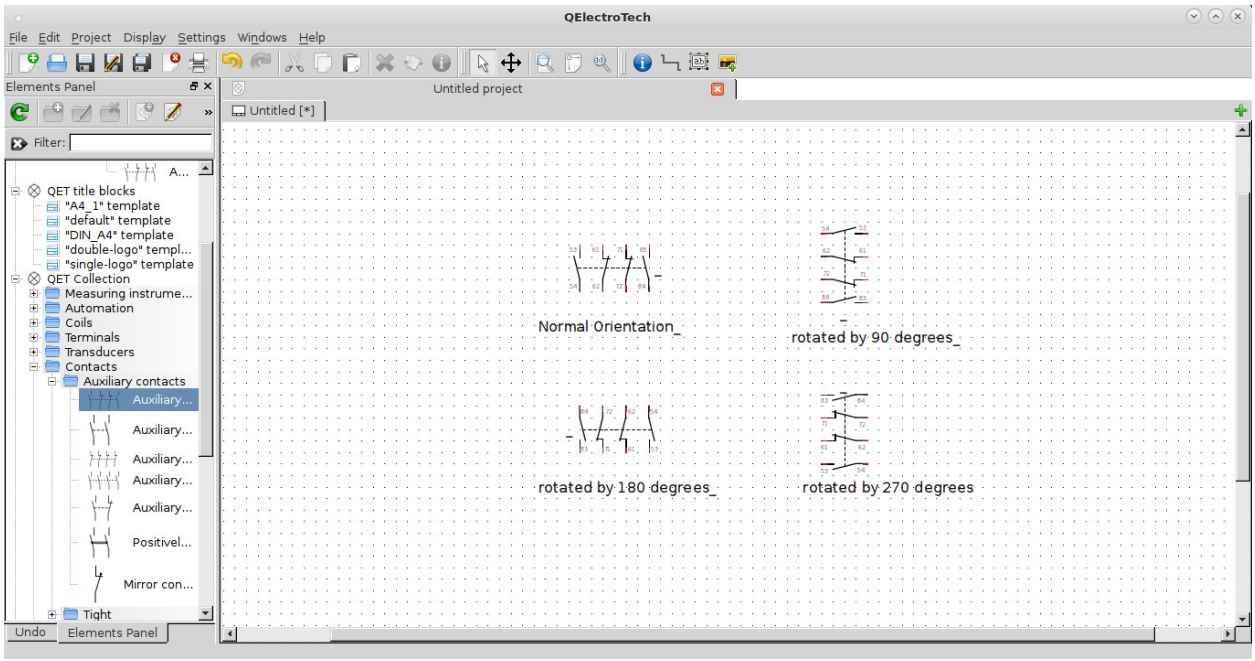
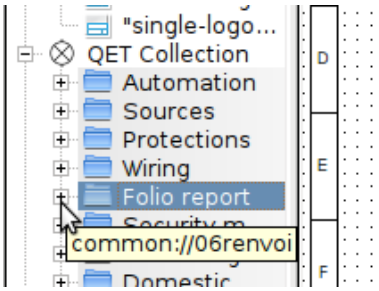


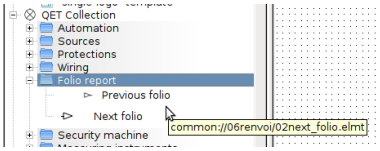
Figure 6: Rotation of an element in work space

6.1 Folio report

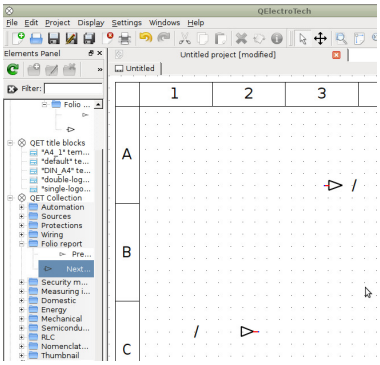
Folio report is included in the 'QET Collection' in collection of QET Elements. The element performs the referencing roles in the drawings. Refer sample drawings enclosed "Ship's electrical generation and distribution" on page 45. The function of the folio report is explained with the following illustrations.



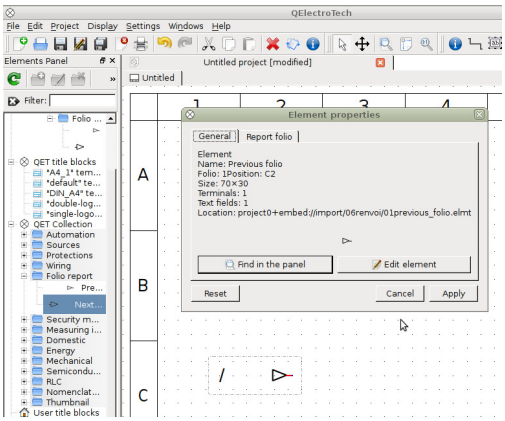
Click '+' of folioreport



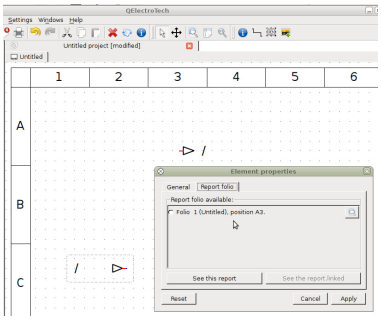
Two elements 'Previous' and 'Next' folio are provided



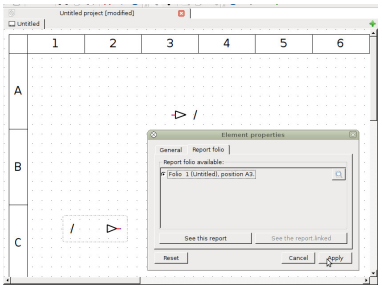
Drag drop each element to a drawing



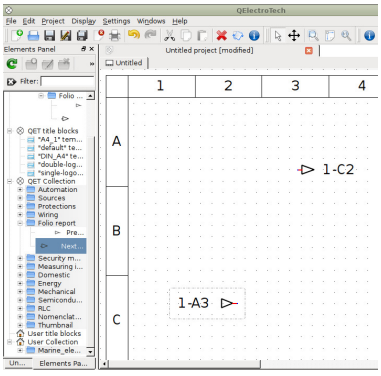
Double click an element for properties window



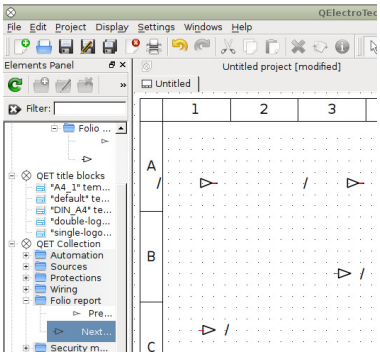
Select report folio tab



Select the radio button in 'report folio available'. Click 'Apply'

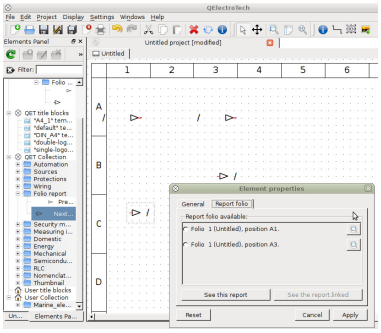


Watch the '\' field of element is updated to text

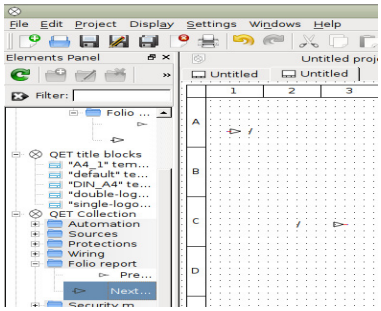


Add each element again as shown to create four elements

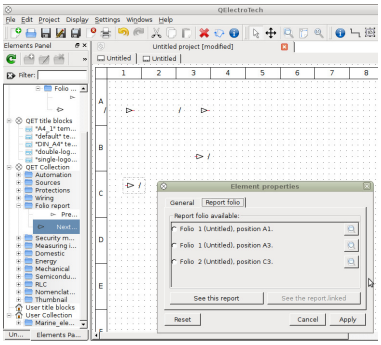
The '\' field of element is updated to text showing that 'from folio 1-A3 cell' and 'to folio 1-C2 cell'. The report folio elements perform referencing roles. This kind of referencing is widely used in electrical and process drawings.



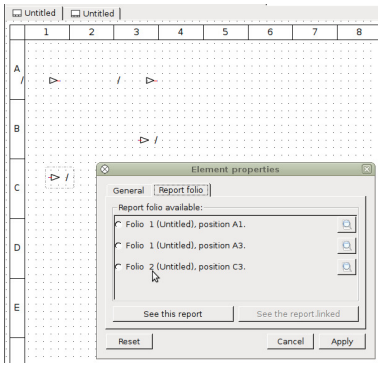
Double click any element to watch its properties window
Two options are now available in 'report folio available'



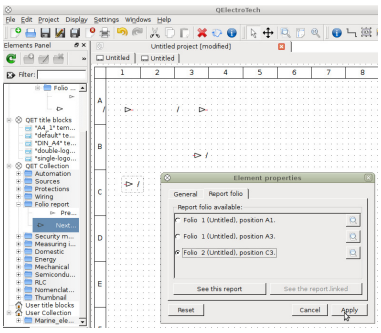
Add a new drawing to the project
and add the two report folio elements to it



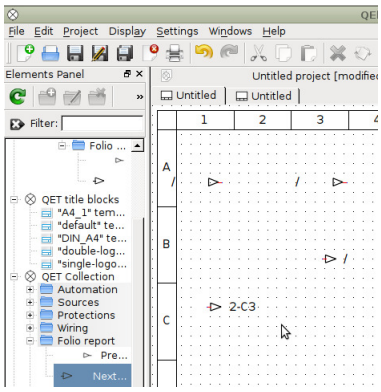
Switch back to first drawing and double click an element
Three options are now available in 'report folio available'



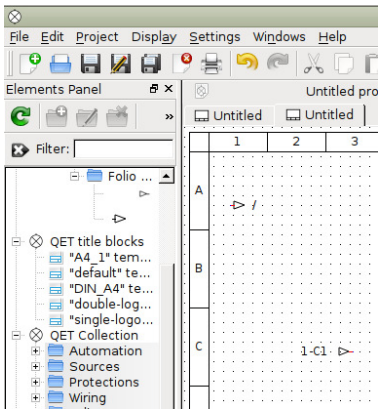
Observe the new element from another drawing
(Folio 2) is available for referencing



Select the folio 2- reference and click 'Apply'



The element updates to 'to folio 2 - cell C3' reference



Switch back to second drawing and observe
The previous folio element is updated to show 'from folio 1 - cell C1'

7 Text field

7.1 Inserting text field

Text field appears like an underscore and it can be either used in association with an element or as an additional field inserted in the work space. Some elements are provided with a text field at each of their terminals and one for the entire element. Additional text fields can be entered any where in the work space by left clicking the 'Add a text field' short cut icon from the tool bar (this action depresses the shortcut text icon in the tool bar) and left clicking the location in the work space where you want to include a text field. The text field can be edited in the work space by simply double left clicking it with the mouse and keying in text from key board and deselecting the text field by left clicking any where else in the work space. The text field supports multi-line entry of text.

7.2 Orientation of the text field in work space

By default all the text fields are horizontally oriented in the work space. They can be oriented in any direction in the work space and are provided with two methods of applying rotation to the field.

1. Selecting the text field by left clicking it with the mouse and rotating it with the 'Rotate' shortcut provided in the tool bar. Alternatively, the text field can also be rotated by left clicking it with the mouse to select it and pressing space bar to achieve desired orientation. This action rotates the text field in quantum steps of 90°.
2. Left clicking the text field with mouse to select it and pressing 'control + space' bar on keyboard pops up a GUI window to orient text filed in any direction, with the precision of two decimal places. The GUI window also facilitates rotation in quantum steps of 45° by clicking the green dots on the yellow circle. A double spin box is also provided to manually enter an angle of orientation upto two decimal places. The double spin box can also be interacted with left mouse click on the up or down arrow keys to increment or decrement angles in steps of 1°.

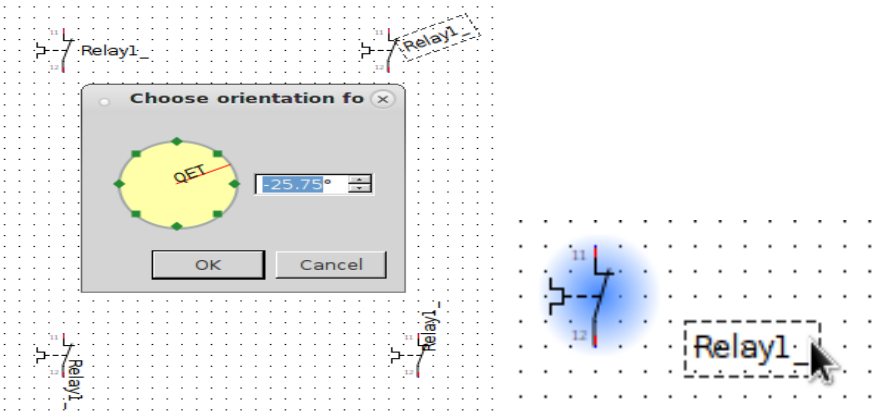


Figure 7: [Left] Text rotate gui window ('control + space') & [Right] Blue shade while moving text

7.3 Moving and editing text field in work space

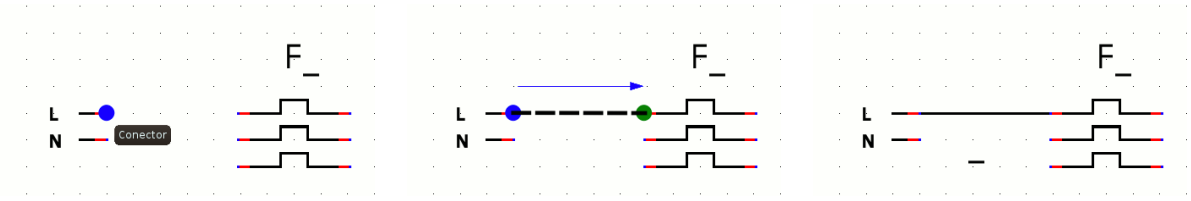
Text field can be moved any where in the workspace by left clicking it with mouse and dragging it to a new place and releasing the mouse click. Text fields that are associated with elements require 'control key' to be pressed from the keyboard to permit moving its text field. Moving an element's text field produces a blue colored highlight over the element during relocation as shown in the Figure 7 (right). This is a feature of QElectroTech to help user perform such actions in a large diagram containing a lot of elements.
Text fields that are not associated with elements have an additional editing option accessible by selecting the text field with a left mouse click and using the keyboard shortcut 'Control+e' or from main menu bar, 'Edit —> Edit the text field'. A simple text editor window opens up to facilitate increasing the font size, font color, applying bold / italics / underlining texts and inserting a hyperlink. The hyperlink is resourceful for diagrams printed to 'pdf' format.

Document and graphics created under Creative Commons Attribution 3.0 License

8 Working with connectors

Elements have terminals, which facilitate connecting them with other elements with the help of connectors. Making connections between terminals can be summarized as follows:

1. Position the cursor on the element connector or terminal you want to connect; you will see that a blue dot develops on the terminals you want to join.
2. Left click the blue dot and hold and drag the pointer to the connector or terminal to join the other element you want to connect. If a green dot appears on the target terminals, it means that the union is complete. A red dot means a “forbidden connection”.
3. Release the left mouse button and the line (left) will be completed, if the line does not take the desired position or does not generate acceptable paths, the connectors can be edited by selecting the line and drag from the green markers that form when you position the cursor over the connector line.



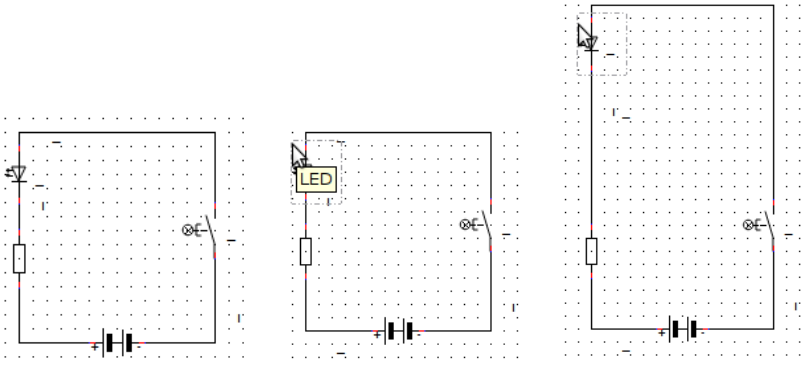
Step 1 **Step 2** **Step 3**

Steps of making connections between terminals

8.1 Re-sizing connectors

Adjusting connectors by moving elements

1. Select an element in a circuit by left clicking it and hold it in the work space.
2. Drag the selected element in the circuit, the connectors linking the element to the rest of the circuit also moves.



Initial circuit **Select an element** **Drag & release**

Adjusting connectors by moving element in a circuit

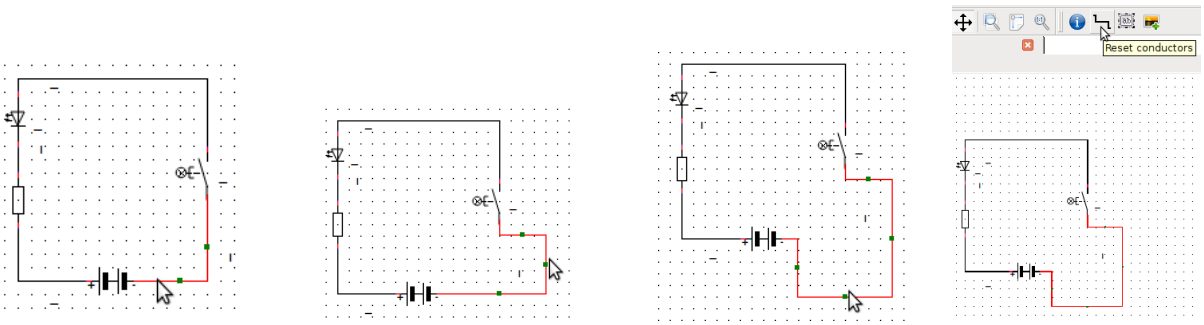
Adjusting connectors with handles

1. Select a connector with a left mouse click. The segment of the selected connector between two elements turns red, indicating that the conductor is selected.

2. Position the cursor over the selected connector; you will find thick green colored squares appear over this segment, one each in a bend.

3. The connector can now be adjusted as per the users demands by left clicking these thick little squares and holding and dragging to a new position. The connector changes its path during this action.

4. To reset the altered path, left click the short cut icon 'Reset conductors' provided in the tool bar with the connector selected (highlighted red). This action will undo all the earlier changes effected to the connector.



Select a connector **Hold a green square and drag** **Drag another connector** **Reset connectors**

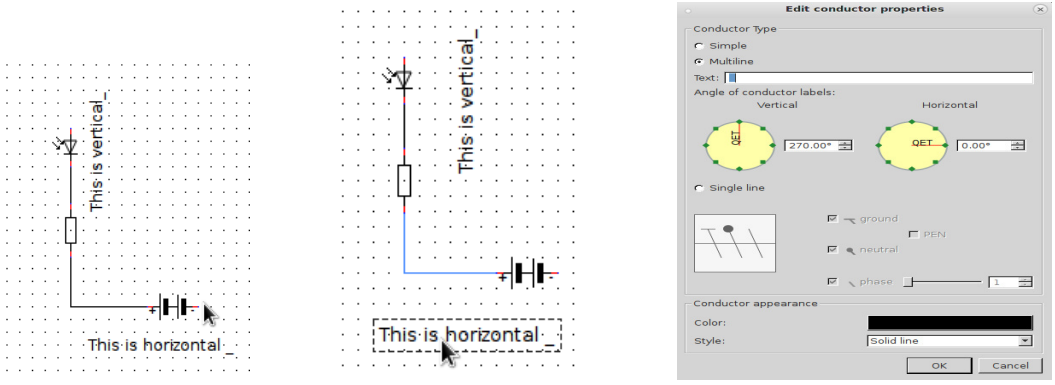
Adjusting connectors in a circuit

Adding text to connectors

Connectors are provided with text fields. Additional text fields can be inserted at desired locations. Text fields for connectors have the same behavior as the text fields for elements discussed under section 7. Double left click a connector text field and enter the text. The text can orient horizontally or vertically depending on the section of the connector where the text field is provided. The text fields can be rotated and re-positioned as required.

Connector properties window

A connector's properties window can be activated by double left clicking it. The connector and its text field properties can be set from this window. Only multiline connectors have text fields. Selecting 'simple' or 'single line' radio buttons deactivates the text field. The 'single line' option has further options to format the connector as an earth, phase or a neutral conductor or a combination of any. Selecting the 'neutral' option further facilitates formatting the conductor as 'Protective Earth Neutral'. Number of phases can also be set upto 3 by selecting the 'phase' radio button and using the slider or keying a value into the double spin field. Color and styles to a connector can be applied irrespective of other choices.



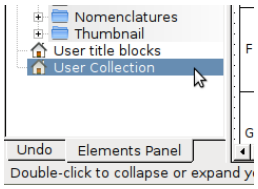
Text orientation **Dragging a connector's text field** **Connector properties window**

Connector's text field & properties window

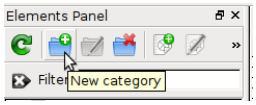
9 Element's editor

9.1 Creating a new element

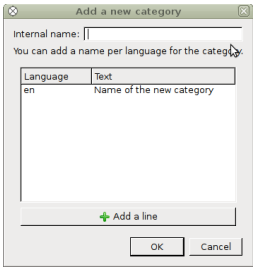
Elements in QElectroTech are stored in 'xml' format. Elements created in Linux distros are usually saved in an invisible folder '.qet'. It is created by QElectroTech in '/home/USER/' directory.



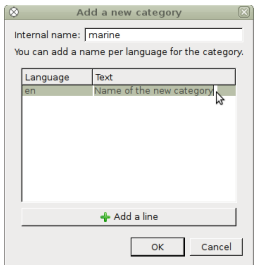
Select 'User collection'
Step 1



click 'New category'
Step 2



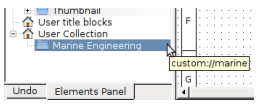
Enter the internal name
Step 3



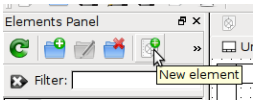
Enter a display name
Step 4

Elements provided in the QET Collection are read only and should not be edited. However, they can be added to 'User collection' and subsequently edited and saved. The above pictures summarize the first four steps in creating a new element.

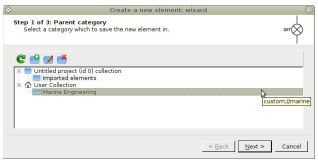
1. Select the 'User collection' with a left mouse click over it. Then elements can directly be created under it. However, it is a good practice to first create a 'category' under 'User collection'.
2. Left click the shortcut icon 'New category'. 'Add a new category' wizard opens up, which guides the user.
3. Enter a name to the new category (internal name); the field takes only small letters, numbers and '-','_' and '.'.
4. Double left click the text field, enter a name that the category should display and hit 'enter' from keyboard. By default, the field shows 'Name of the new category' and language 'en' for English versions. Additional support languages can also be added by left clicking the 'Add a line' button. Now left click 'OK' button to add the category to user collection. This action can also be achieved by hitting 'enter' key from keyboard.



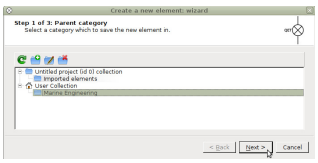
New category added
Step 5



click 'New element'
Step 6



Select the category
Step 7



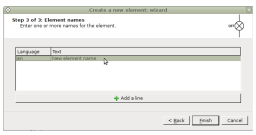
Left click on 'Next'
Step 8

A new category is now added under 'User collection'; its internal name is displayed in the tool tip and the name displayed is from the text entered in the text field in step 4.

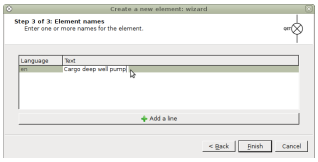
1. Select this category with left mouse click.
2. Click on the shortcut icon on tool bar to create a 'New element'.
3. A 'Create a new element' wizard opens up, which guides the user to create an element. Select the category in which the new element created will be placed.
4. Click 'Next' button.

Document and graphics created under Creative Commons Attribution 3.0 License

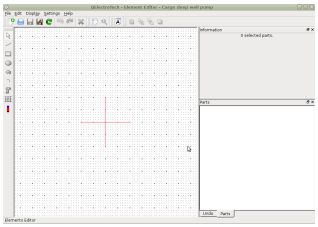
Enter an internal name
Step 9



Select the text to display
Step 10



Enter the text to display
Step 11



Element editor opens
Step 12

Step 8 opens up a request for a file name for the element to be created.

1. Enter a name for the element to be created. Naming convention is similar to that of the internal name for category explained earlier in section 9.1. Click 'Next' to continue the wizard.
2. Select the text field and left double click on the default text displayed. The field will be editable now.
3. Enter a name by which the element will be displayed under 'User collection'. The text field accepts all valid keys. After entering a name, hit 'Enter' from key board. Hit 'Enter' key from key board one more time or click 'Finish' button to complete the wizard.
4. The wizard now completes opening up 'Element editor' window. The Element editor facilitates drawing of a new element or editing imported elements under 'User collection'. The element editor has a plain drawing area with two thin red colored reference cross hairs. The cross hair is for reference and does not show up in the finished element sketch when saved.

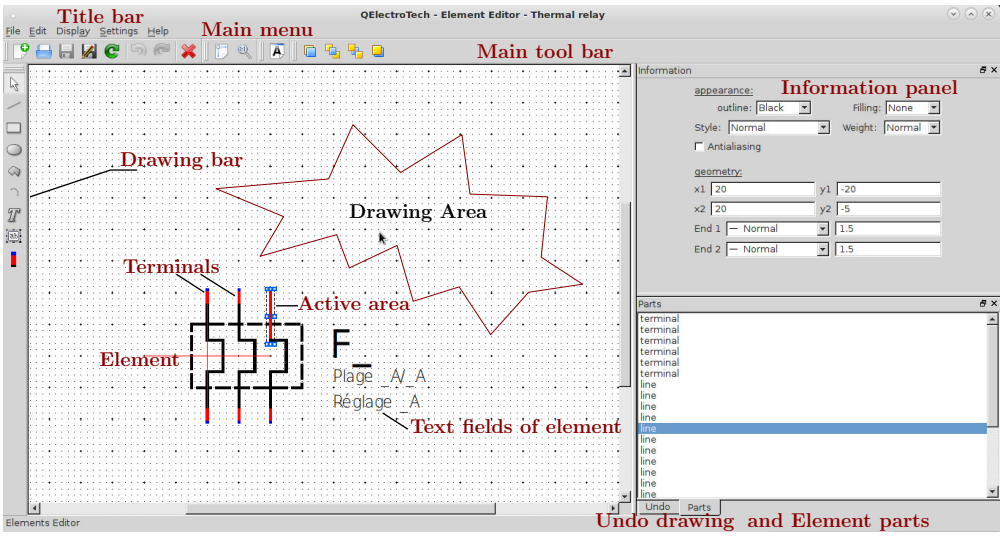


Figure 8: Element editor window

9.2 Description of Element Editor

Title bar:

The title bar has buttons that all standard windows are provided with.

Drawing area:

This is the dotted grid over which the elements are drawn.

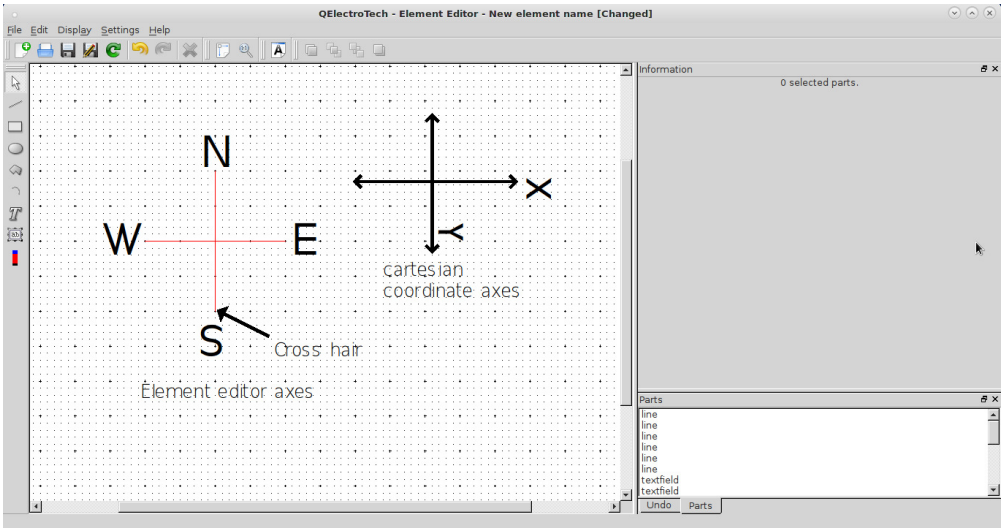


Figure 9: Comparison of Cartesian coordinates (Mathematics notation) with Element editor field. (The letters 'x' and 'y' show positive direction of axes in Cartesian field).

Undo & Parts:

By default the element's 'parts' is the active tab. It contains the all the individual segments that make up an element in the chronological order. The 'Active area' segment is highlighted in this tab. Undo tab records each action made in the element editor and the user can visit any stage by selecting the corresponding entry in this tab.

Main menu:

Main menu has 'File', 'Edit', 'Display', 'Settings' and 'Help' icons.

File: The file option has standard windows options for creating a new element, opening a saved '*.elmt' file or '*.xml' drawings, opening an element from 'User collection' or 'imported collection', saving the drawing to their respective locations and inter alia. 'Reload' option reloads the element drawing neglecting all the changes effected.

Edit: 'Edit' has standard options like 'cut', 'copy', 'paste', 'select all', 'undo', 'redo' and 'delete'. 'Invert selection' selects all geometry in the drawing area other than the 'active area'. There is an option to paste a drawing or an element from a saved '*.xml' or *.elmt' format file to the existing drawing. A file location or an element from 'User collection' or 'imported collection' can also be selected. The 'Edit' also has special commands 'raise', 'lower', 'bring forward' and 'send backward' to adduce to overlapping geometry. The order sets priority as to which geometry should overlap the other in the drawing area. It has analogy similar to shuffling a pack of playing cards. Options to set the author name and his license and adding language features to the drawing are also available.

Display: This icon has the zoom in / zoom out options for the drawing. Settings: Settings has features to turn on or off panels or tabs that constitute the drawing editor, a full screen mode and configure QElectroTech options. Configuring QElectroTech will be described in a later section.

Information panel:

It is also the properties window for the selected individual segments that make up an element. The information panel contains two groups of properties 'appearance' and 'geometry'. Geometry defines the segment selected in the work area (active area). It is possible to fine tune the coordinates of the segment from the information panel. The geometry describes the'active area' selected. A simple line, a square or a rectangle, a circle, text fields, a terminal and an arc will have their own set of specific parameters, which are displayed in the information panel. Try drawing each of the drawing tools in the work area and select them to define 'active area' and check the information area. Watch how the parameters change with each geometry; also try changing the parameters to note their effect on the 'active area'. The appearance properties define the line style, color, weight, filling for closed geometry like rectangle,

square etc., and another property called 'Anti-Aliasing', which is to remove distortions of the sketches and smoothen them for better appearance. Geometry parameters define the 'active area' and they are shape specific; for example, a line segment will have start and finish coordinates, beginning side or ending styles (to draw arrows, diamonds etc.,) and corresponding weight factor to size these ending styles.

Active area:

Active area is the part of the element that is selected with a left mouse click. The active area data is displayed in the information panel and the segment will be highlighted in the 'element parts' of the editor.

Element:

Element is the completed drawing of a symbol, which is used to represent a component in a circuit drawing. Elements can be given names to describe them and saved either in a *.elmt or a *.xml format. An element should be provided with at least one terminal to facilitate its integration in a drawing. The element can also be saved to 'User collection' among others. Two types of text fields are possible to be added to an element to either describe it or its segments and terminals. The text fields are described in detail later under section 9.2. One text field is not editable when the element is saved and used in the QElectroTech drawing while the other has users text input feature.

Main tool bar:

The main tool bar is a collection of quickly accessible shortcuts to the features available in 'Main menu' group under 'File', 'Edit' and 'Display'.

Drawing bar:

The drawing bar has the tools for constructing an element. The options include a cursor tool to permit selection of either individual segments of the drawing or a combination of these, including text and terminals. The drawing tools are each described below:

Line tool: Use a left mouse click to select and activate the 'Add a line' tool from the drawing bar. Use a single left click in the drawing area to select a coordinate and hold and drag release to another coordinate to define the line segment. The line segment can be re-sized either from its information panel or using the sizing handles from its 'active area'. Drag dropping a line segment to another location in the drawing is also possible.

The line segment information is shown here in Figure 9. The line segment geometry can be defined by a start coordinate and ending coordinate. The default end style is 'normal'; options to set this style to 'simple arrow', 'triangle arrow', 'circle arrow' and 'diamond arrow' are available. End 1 is the initial point from where the line segment is drawn and End 2 is the ending point of the line segment. Triangle, circle and diamond arrow spaces can be filled with a color using the 'Filling' combo box. The line segment can be colored 'white', 'red', 'green' and 'blue' apart from default red color. The line style can have 'normal', 'dashed', 'dotted' or 'dashed & dotted' styles. The thickness of the line segment can be defined from the 'weight' combo box. A slanted line can have rough outline which can be smoothed by selecting the 'anti-aliasing' option.

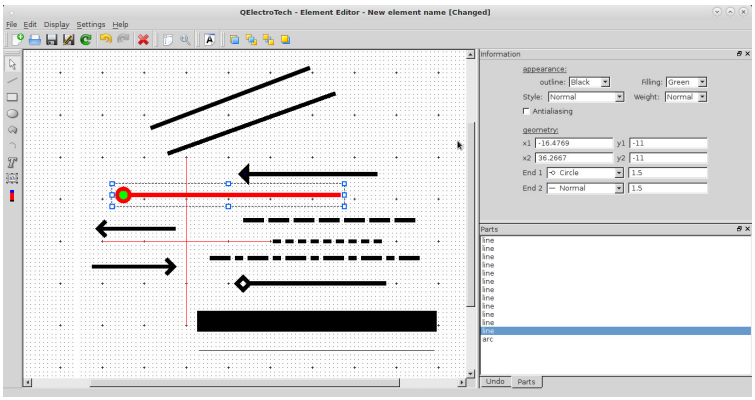


Figure 10: Line tool and information

Rectangle tool: Select 'Add a rectangle' icon with a left mouse click from the drawing bar to activate it. Use a left mouse click to select a point in the drawing area and hold and drag to another coordinate and release the click to complete a rectangle. The geometry can be re-sized from its information panel or by using the resizing

handles from its 'active area'. Drag dropping the rectangle to another position in the drawing area is also possible.

The Rectangle tool permits drawing of rectangular geometric sketches in the element editor. Some of the formatting that can be done from the information window to a rectangle is shown in the Figure 10. The geometry is defined by a point and the size of the rectangle (length and breadth). The appearance options are the same as available for line tool.

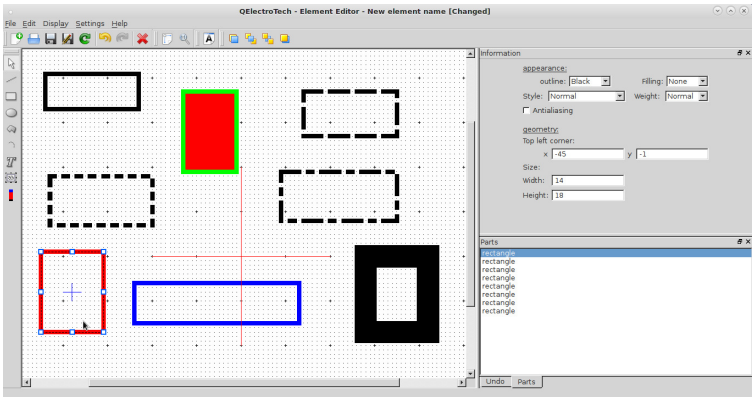


Figure 11: Rectangle tool and information

Terminal tool: Terminals have to be provided for the element to integrate it in a circuit. The element editor will not save a drawing without at least one terminal. Terminals can be drawn from the short red-blue line icon provided in the drawing bar. It is described in the Figure 11. Left mouse click on the 'Add a terminal tool' from the drawing bar to activate it and left click on a point in the drawing area where you want to place this terminal. Once added it can be selected and drag dropped to a suitable location.

Terminal is a not a simple drawing tool like 'line' or 'circle'. It is a tool which gives an element a scope to be connected to other elements in a circuit. Terminal is mandatory for each element, more terminals can be provided depending on the type of element being drawn. It appears as a line in two distinct colors 'blue' and 'red'. It has fixed dimensions and cannot be changed. It has a special directional property by way of 'North', 'South', 'East' and 'West', which can be changed from the information panel, when the terminal is the 'active area'. This direction is determined by the blue end of the terminal; in the direction that it points. It is described in the working area by a single coordinate, the point where it is added. The red end of the terminal joins to the element geometry. The terminal gives an element a dynamic property in QElectroTech. When you point cursor to an element terminal after it is added in a drawing, the blue end turns to a big blue dot, indicating its intention to be connected to a point in the circuit.

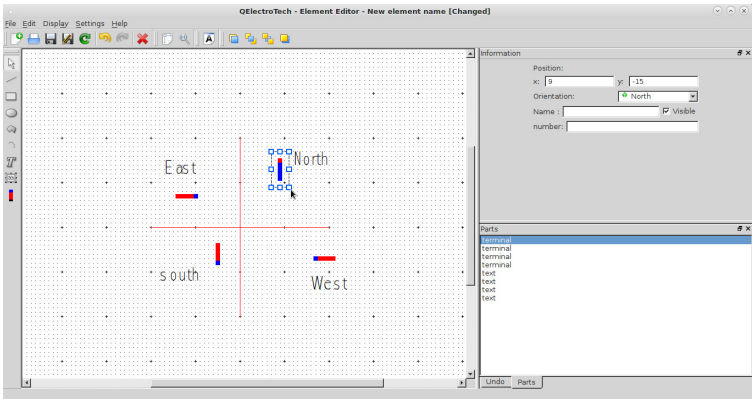


Figure 12: Terminal tool and information

Ellipse tool: Select the 'Add an ellipse' icon from the drawing bar to activate it. Use a left mouse click in the drawing area to select a point and hold and drag to another coordinate and release the click to form a geometry. You can re-size the geometry developed to your desired intention either by using re-sizing handles of its 'active area' or using its information panel. Drag dropping of the geometry to a desired position after selecting it is possible.

Polygon tool: Select the 'Add a polygon' icon with a left mouse click to activate the tool. With the tool activated, use left mouse clicks to select a number of points that define your geometry in the 'drawing area'. To finalize the

Ellipse tool permits drawing ellipses and circles in the drawing editor. The geometry of an ellipse is defined by point coordinate and its horizontal and vertical diameters. Standard line formatting and styles are possible with ellipse tool. Some of them are illustrated in the Figure 12. Anti-Aliasing is a default option with ellipse tool, which makes its appearance smooth. This option is deselected for some of the illustrations shown in Figure 12, can you identify these ? ☺

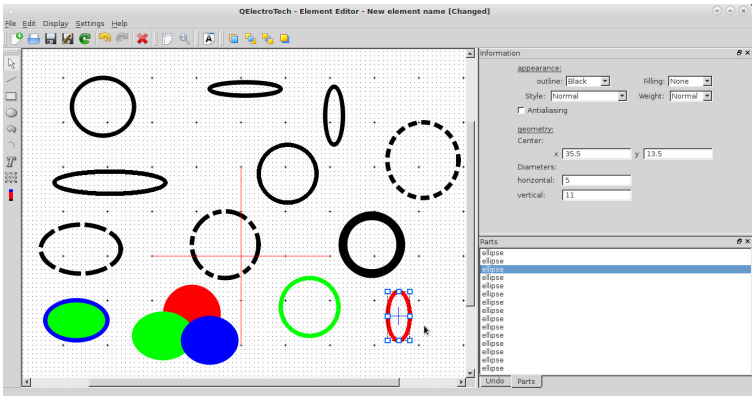


Figure 13: Ellipse tool and information

geometry, use a right mouse click. You can re-size the geometry using the handles of its 'active area' or from its information panel.

'Add a polygon' tool is essentially a utility tool, which permits creation of varied geometry. It is a handy tool for creative users trying to create complicated symbols. Some samples are drawn in Figure 13 to display some of its abilities. The geometry of a polygon is defined by a set of coordinates, which are created by left mouse clicks in the drawing area. The appearance properties are same as that of a line tool. There is an additional option for polygon tool in the information panel, a 'Closed polygon' selection button. If the button is selected with a sketch as 'active area' drawn with the polygon tool, a closed geometry is developed. This action permits further formatting by the 'Filling' combo box in the appearance section of the information panel. Only the closed polygons accept filling with colors.

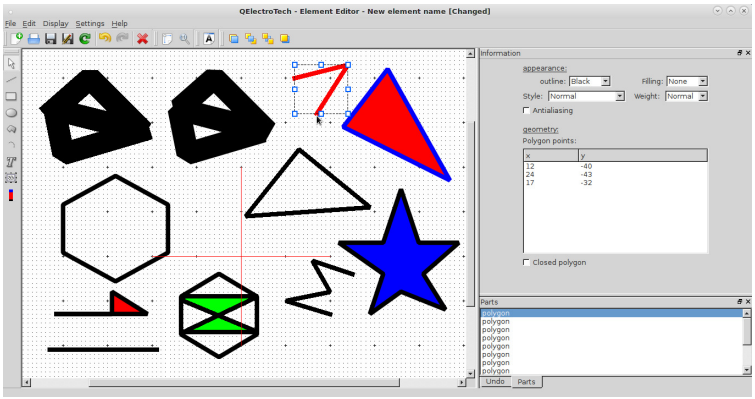


Figure 14: Polygon tool and information

Arc tool: Select the 'Add an arc' icon with a left mouse click to activate the tool in the drawing bar. With the tool activated, use left mouse click to select a point in the 'drawing area' and hold and drag to a new position and release the click. The new point can be described at a distance of horizontal diameter along x-axis and a vertical diameter distance along y-axis from the initial point. You can re-size the geometry using the handles of its 'active area' or from its information panel. Arc tool also has anti-aliasing as its default option for smoothness.

Add Text: Select the 'Add text' icon with a left mouse click to activate the tool in the drawing bar. With the tool activated, use left mouse click to select a point in the 'drawing area'. A text box with default font size and orientation appears at the point selected. You can re-size the geometry using the handles of its 'active area' or from its information panel. The add text field is used to label the element and its components. The text added with this tool cannot be edited while using the element in QElectroTech drawings.

Add a text field: Select the 'Add a text field' icon with a left mouse click to activate the tool in the drawing bar. With the tool activated, use left mouse click to select a point in the 'drawing area'. A text box with default font size and orientation appears at the point selected. You can re-size the geometry using the handles of its 'active area' or from its information panel. The add text field is used to label the element and its components. A default '_' is added to this text field to give an indication of the location of the field when the element is added in a QElectroTech drawing. This field accepts the users input in the QElectroTech drawing.

Document and graphics created under Creative Commons Attribution 3.0 License

The 'Add arc' tool permits creation of an arc along a circle's circumference or in an elliptical path. Some samples using arc tool are drawn in Figure 14. The geometry of an arc created by this tool is defined by a coordinate, which is the initial point selected by a left mouse click in the drawing area, a horizontal diameter and a vertical diameter defined as distances along x and y axes from its initial point, a start angle and an arc angle. Start angle is the angle from the 'North' line and arc angle is the total angle that the arc subtends at the center of the circle or an ellipse to which it belongs. The appearance properties are same as that of a line tool. The arc can be re-sized either from the handles of its 'active area' or from the information panel. Drag and drop functions are possible and are similar to those described earlier for other tools.

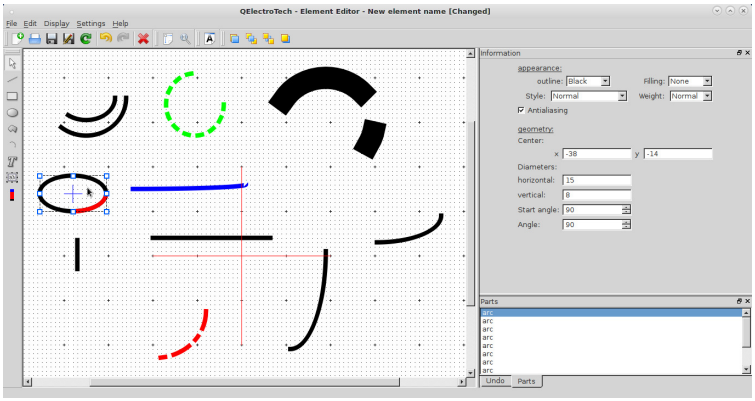


Figure 15: Arc tool and information

'Add text' tool permits fixed naming of the element and its parts at the time of its drawing. Add text appears as a text box with a default font size and orientation at the point selected. The dimensions of the text box can be altered either from the handles of its 'active area' or from the information panel. Drag and drop functions are possible and are similar to those described earlier for other tools. The information panel describes the text box position by a single point coordinate, font size, color as either 'black' or 'white', text to display and orientation. Text can be oriented in any direction possible from 0 to 359.99 °. Figure 15 shows the text added to rectangles filled with different colors.

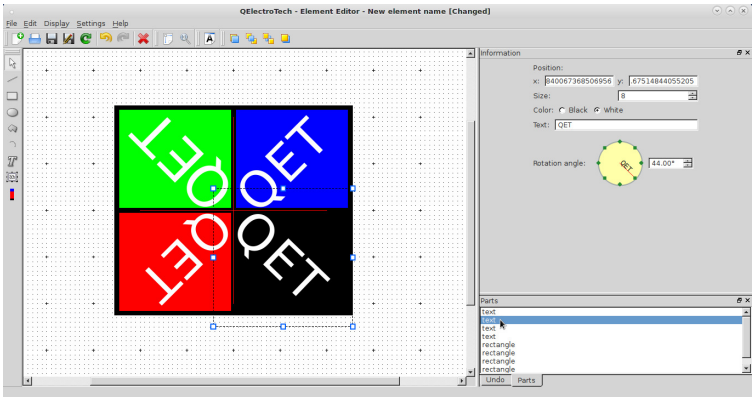


Figure 16: Add text and information

Often 'Add a text field' tool is associated with elements or their parts at the time of its drawing. The text field appears as a text box with a default font size and orientation at the point selected. The dimensions of the text box can be altered either from the handles of its 'active area' or from the information panel. Drag and drop functions are possible and are similar to those described earlier for other tools. The information panel describes the text box position by a single point coordinate, font size, default text as '_' and orientation of 0°. Add a text field can be oriented in any direction possible from 0 to 359.99 °. The Figure 16 shows the Add a text field. This tool has an additional selection box named 'Do not follow parent element rotations'. If this check box is selected, the field does not rotate even if the element to which it is associated is rotated in the QElectroTech drawing.

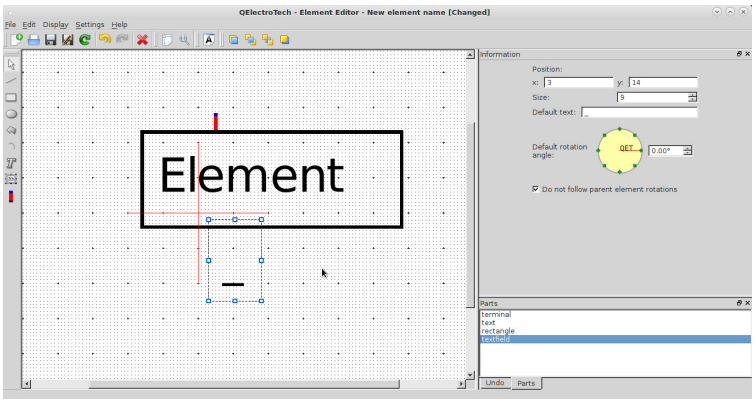
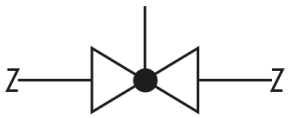


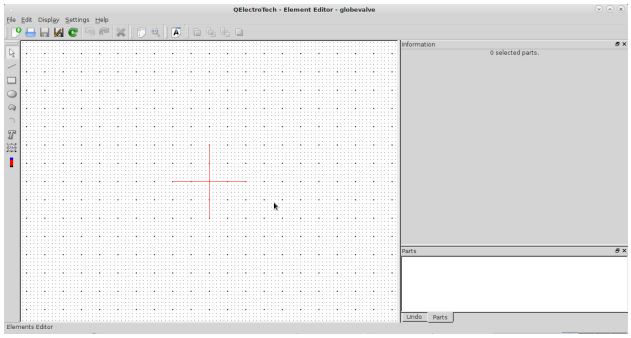
Figure 17: Add a text field and information

10 Sample tutorial - Creating Globe valve element

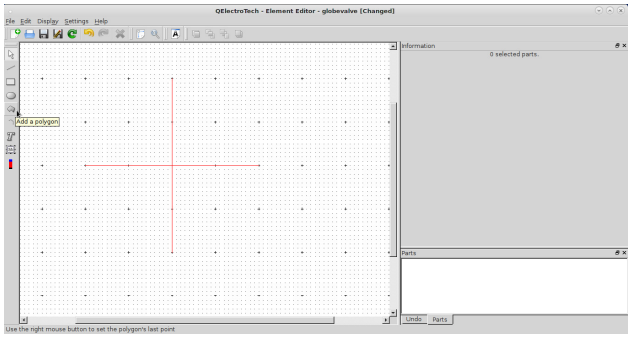
The tutorial here explains the creation of a globe valve element. The final element should look somewhat like this: (**Note:** Steps 34 to 39 demonstrate the functionality of the newly created valve element.)



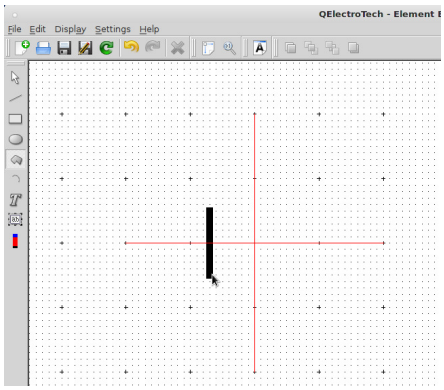
Globe valve



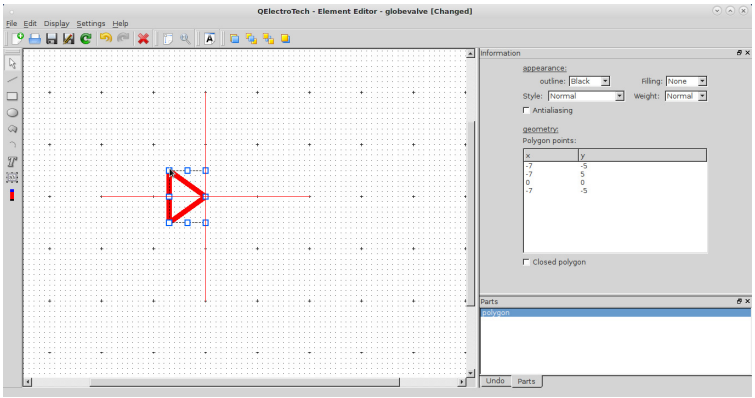
Refer section 9.1 how to create a new element
Step 1



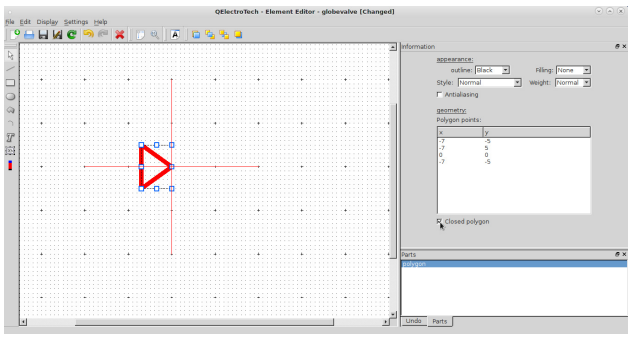
Select add a polygon tool
Step 2



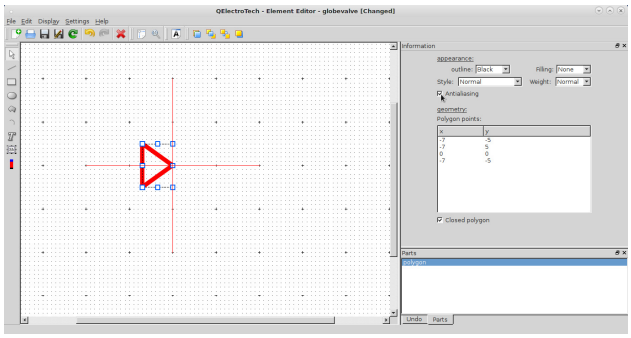
Start constructing a triangle
Step 3



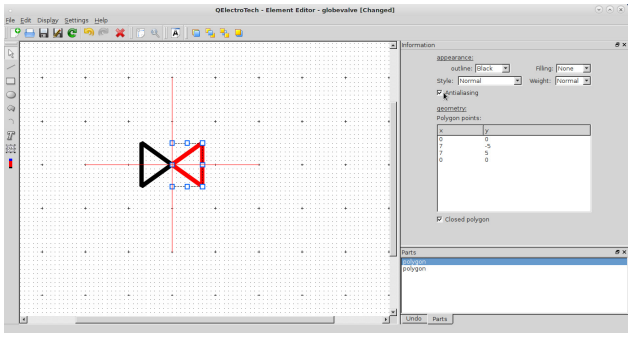
Complete the triangle
Step 4



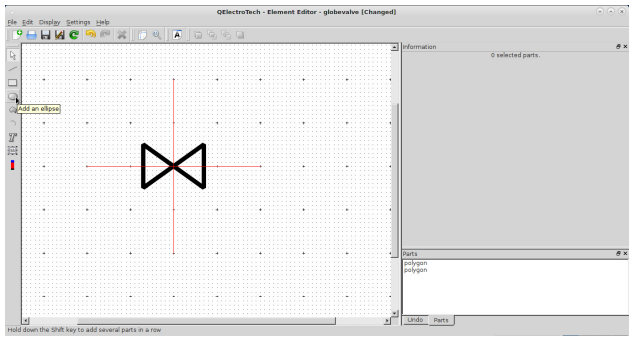
Select close polygon
Step 5



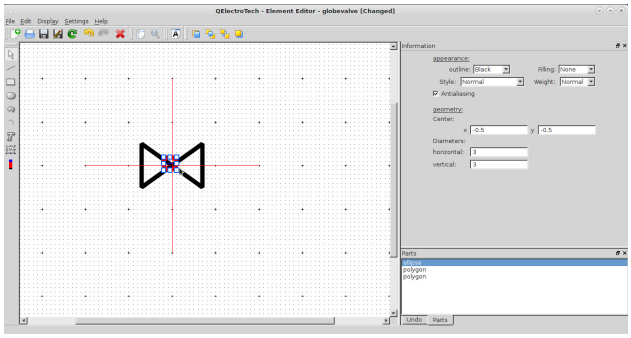
Select anti-aliasing
Step 6



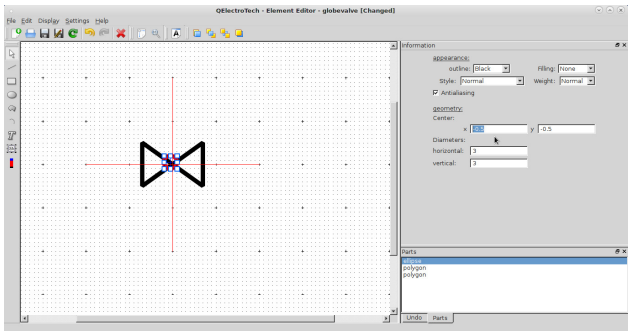
Draw another triangle as shown and select anti-aliasing
Step 7



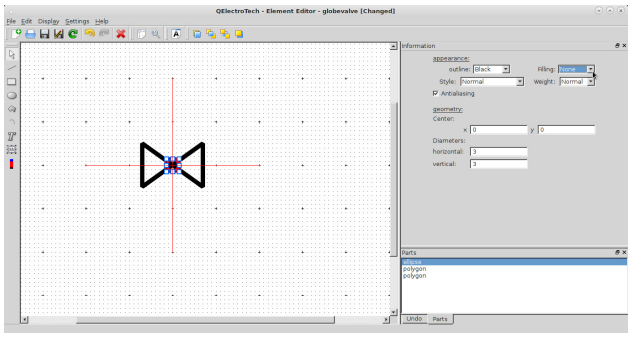
The sketch should now appear as shown. Select ellipse tool
Step 8



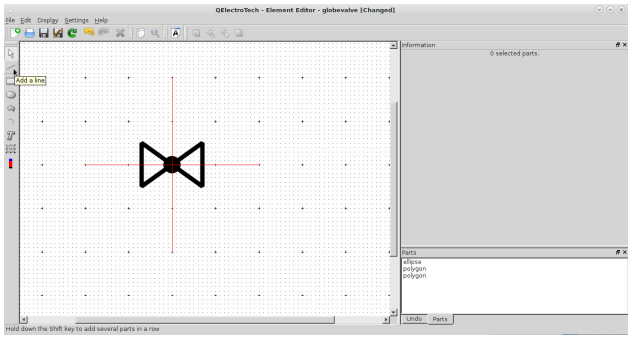
Draw an ellipse near the center
Step 9



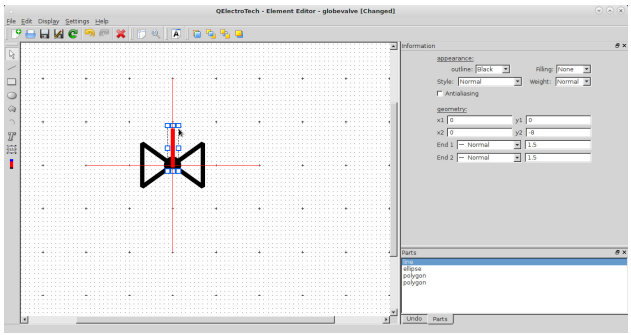
Set its center to origin i.e. (0,0)
Step 10



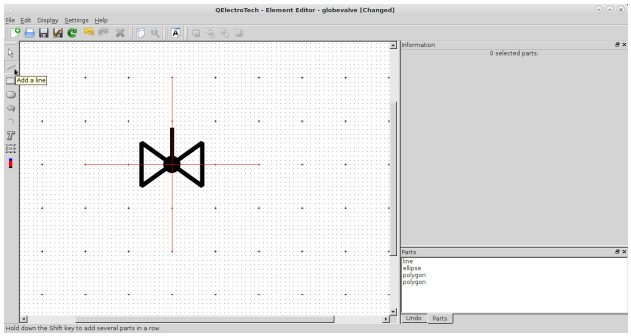
Fill the circle drawn with black color
Step 11



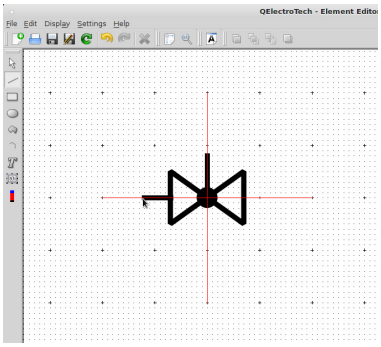
Select the line tool
Step 12



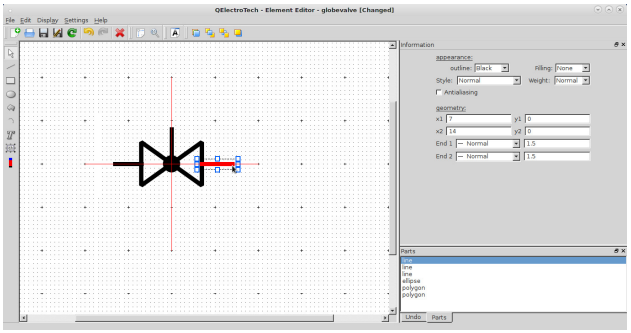
Draw a line segment towards north
Step 13



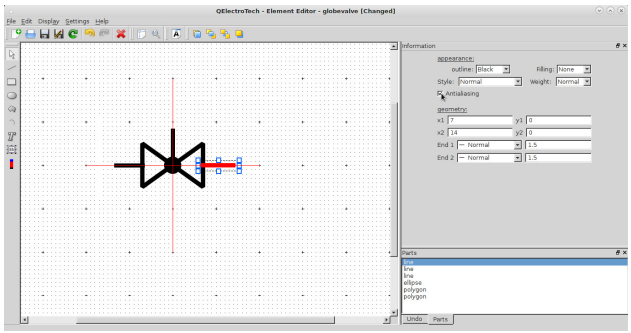
Select the line tool again
Step 14



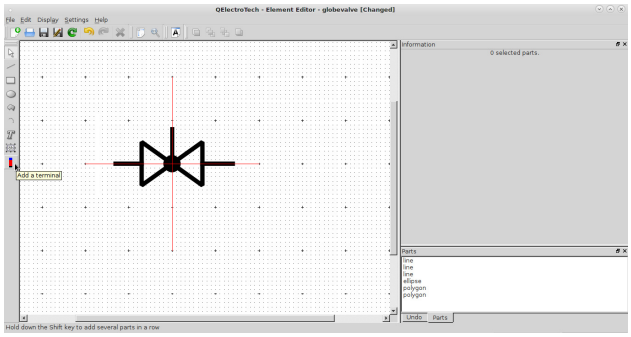
Draw a line segment towards west as shown
Step 15



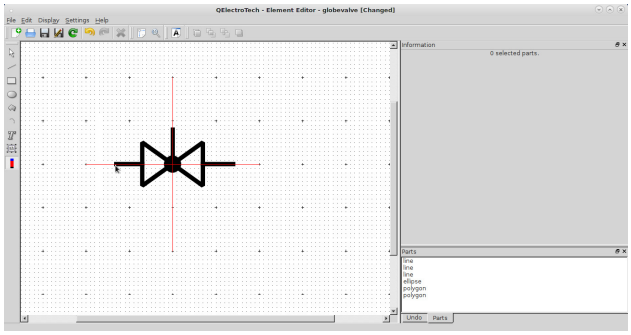
Select the line tool again and draw a line towards east as shown
Step 16



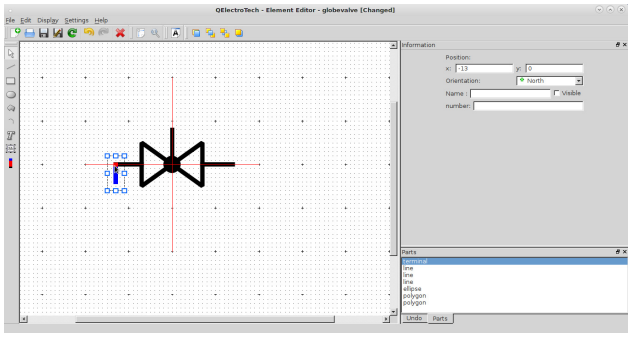
Select anti-aliasing for all line segments drawn
Step 17



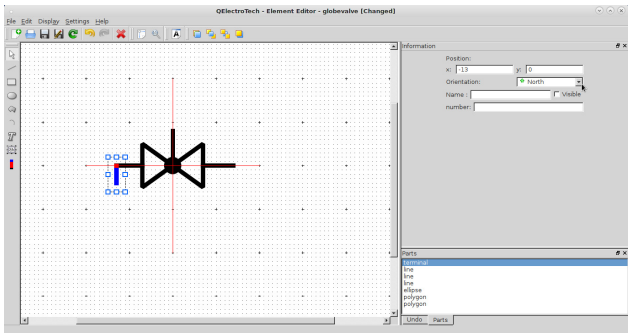
Select the terminal tool
Step 18



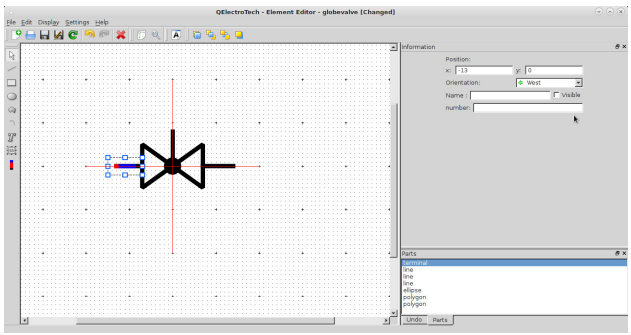
Click on the line segment end to add a terminal
Step 19



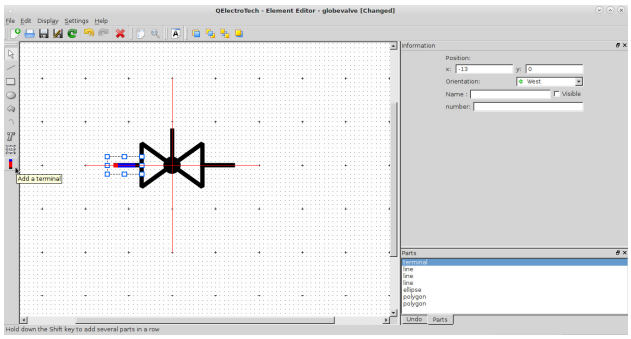
The terminal appears in default orientation (North)
Step 20



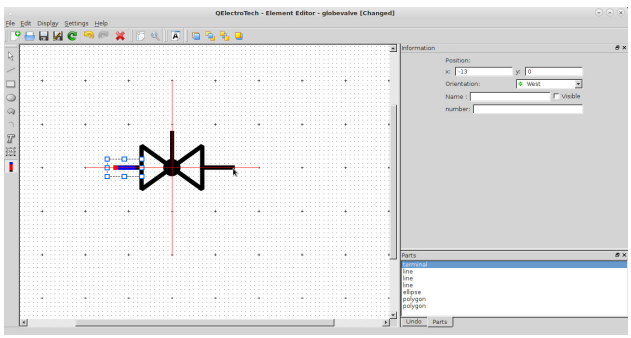
Set the terminal orientation to the west
Step 21



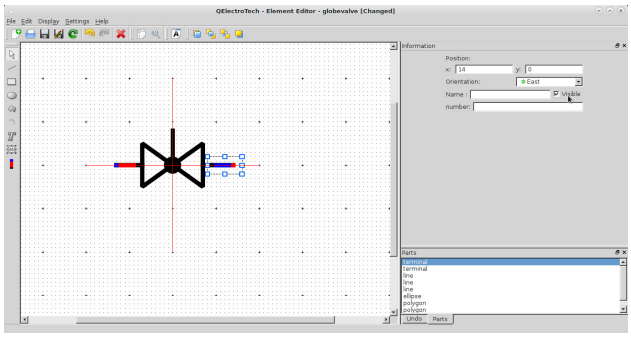
The terminal is now correctly positioned
Step 22



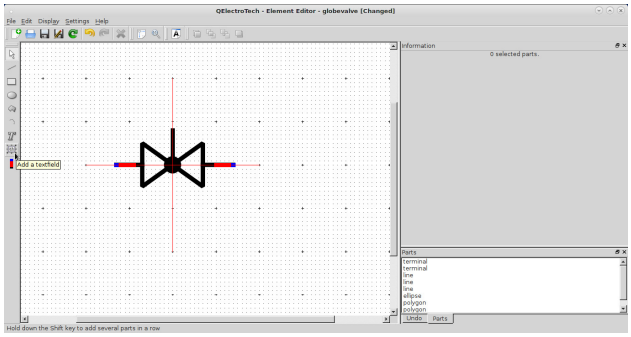
Select the terminal tool again
Step 23



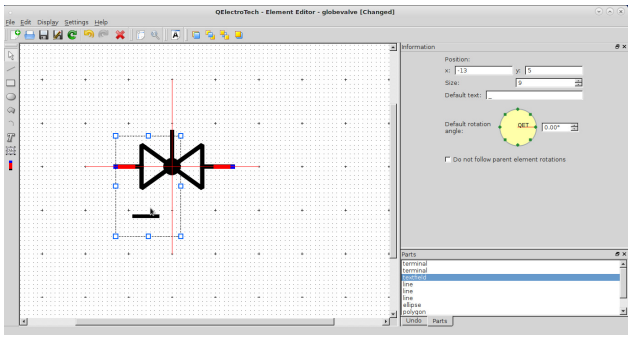
Add the terminal to the other end
Step 24



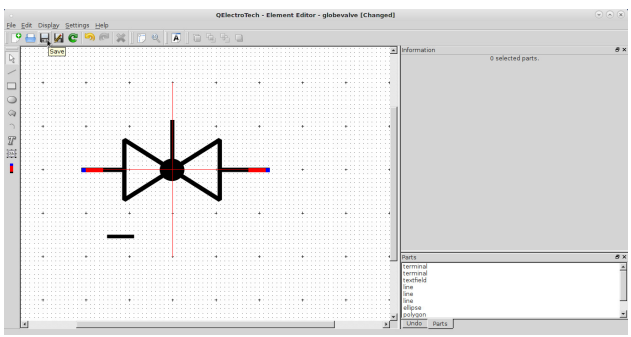
Set the terminal orientation towards east
Step 25



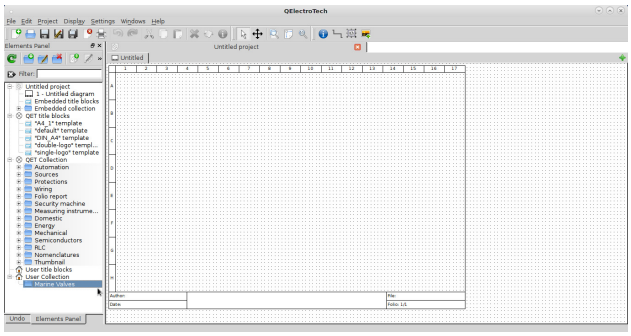
Select the add a text field tool
Step 26



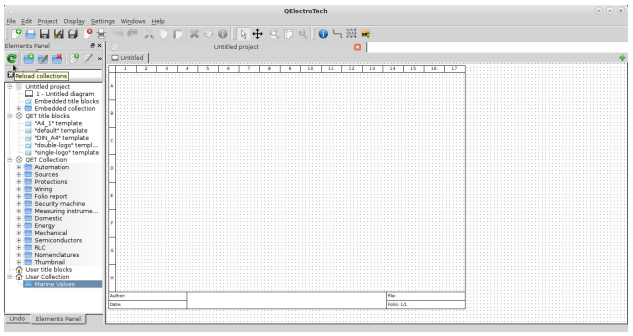
Add the text field to the valve element created
Step 27



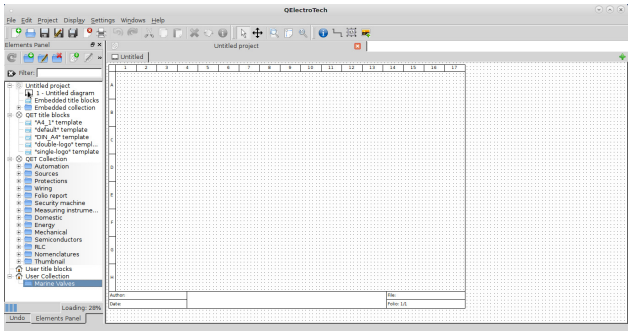
Click the save button to save to 'User collection'
Step 28



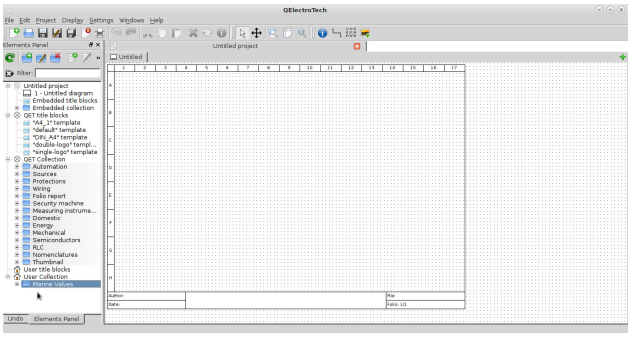
Open the QElectroTech main window
Step 29



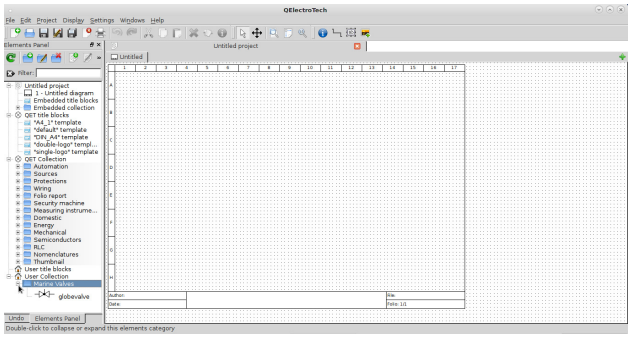
Select the reload button
Step 30



Wait for the new element to appear in a category created under 'User collection'
Step 31

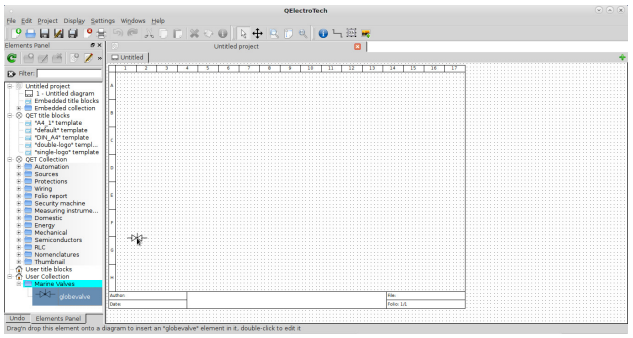


The category under 'User collection' gets updated to a tree '+' appears for the category
Step 32

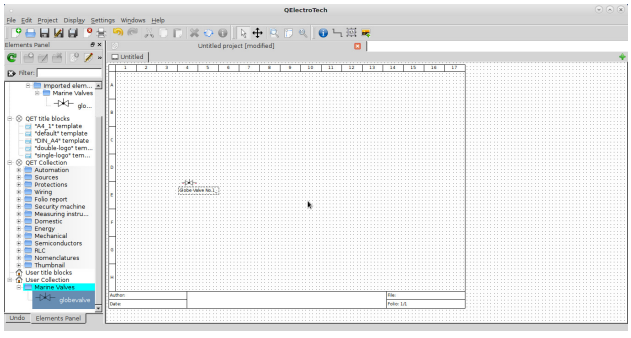


Click on the '+' sign to view the element added

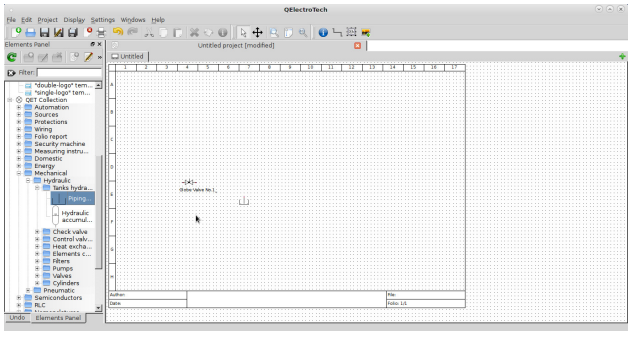
Step 33



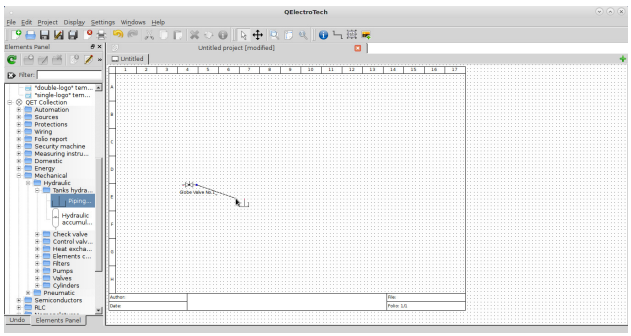
Select the element, hold and drag drop into drawing area
Step 34



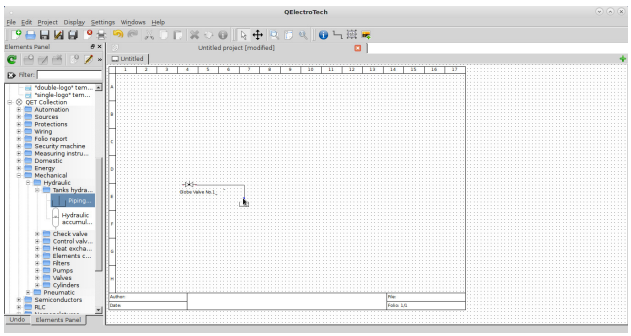
Give a description in the text field
Step 35



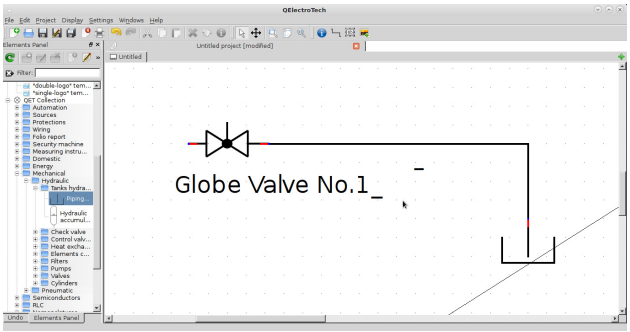
Select the tank element and drag drop it to drawing area
Step 36



Click on the valve element terminal, hold drag to the tank terminal
Step 37



The new valve element is connected to the tank, demonstrating functionality
Step 38

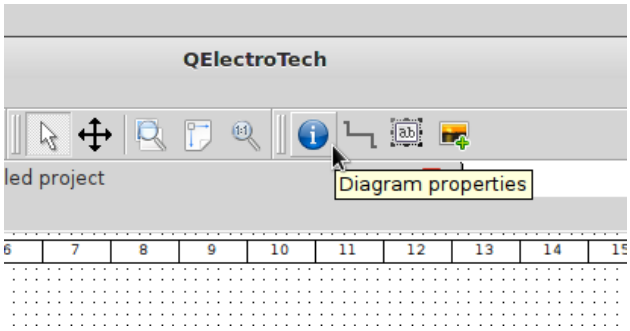


Click 'zoom content' on the tool bar for a closer view
Step 39

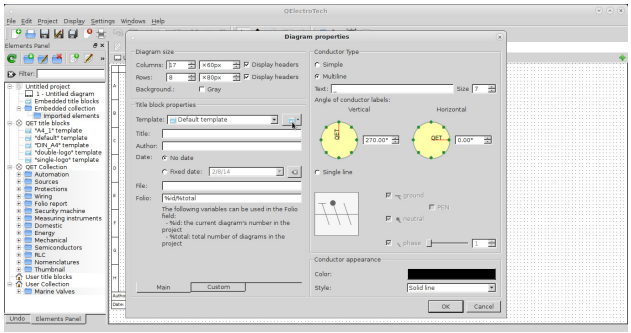
11 Title block template editor

QElectroTech comes with five different templates to work with, namely - ‘A4_1’, ‘default’, ‘DIN_A4’, ‘double-logo’ and ‘single logo’. QElectroTech also permits custom template designing with user defined title block. In this section an introduction to the title block editor is presented. Also, a tutorial to design a referenced title block is included, to familiarize users with the behavior of the title block editor.

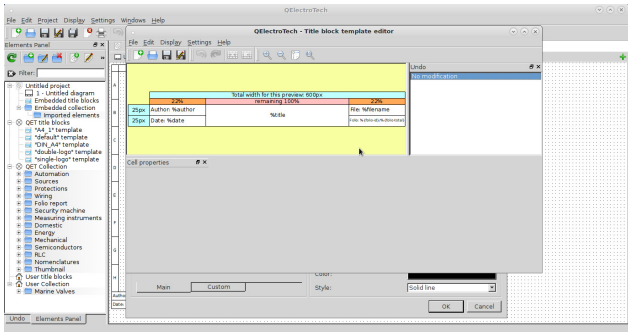
11.1 Opening title block editor



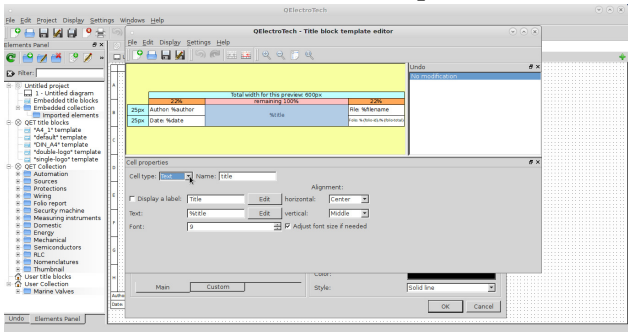
Click on the diagram properties



Click the button to the left of the template combo box and select ‘edit this template’



Title block editor opens the title block for editing



There are three types of cells - ‘empty’, ‘text’ and ‘logo’

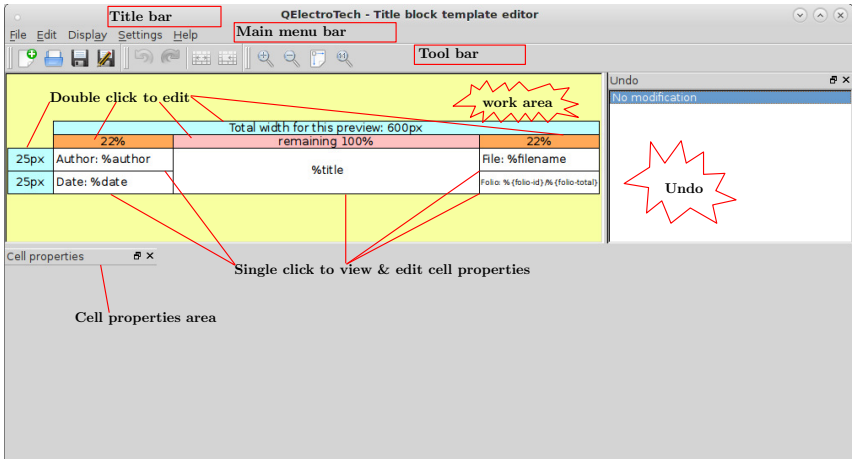
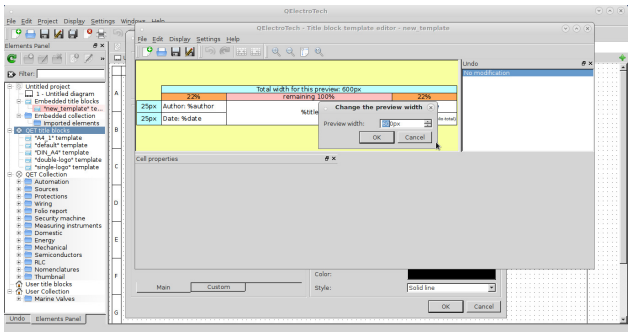


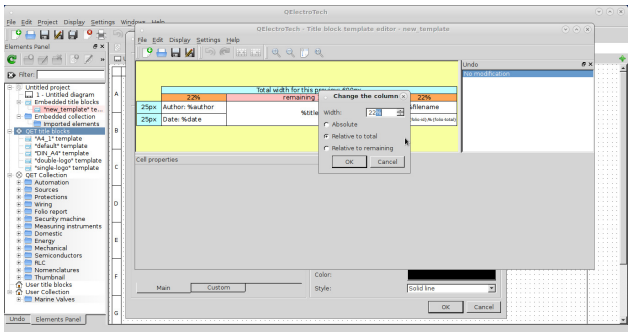
Figure 18: Title block editor

Title block editor has ‘Title bar’,‘Main menu’, ‘Tool bar’, ‘undo’, ‘cell properties’ and ‘work area’ (to edit or design a template)

11.2 Title block editor features

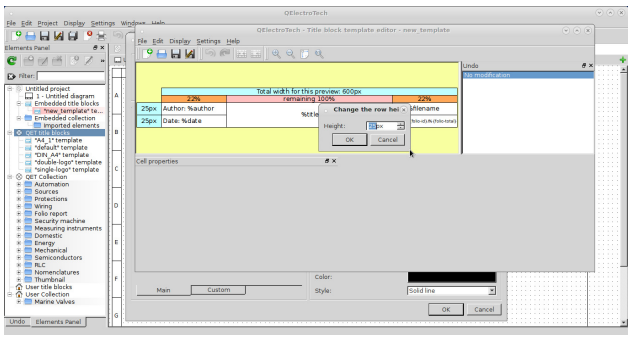


Double click total width preview cell to adjust

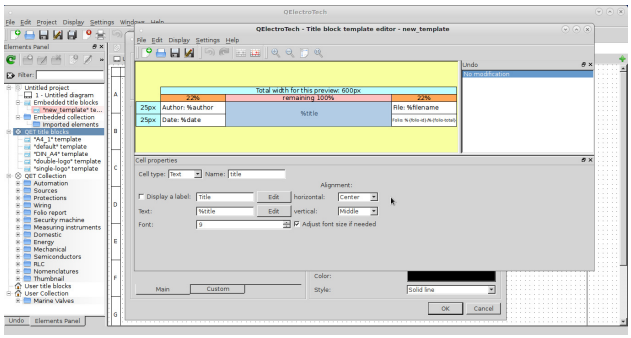


Double click column width cells

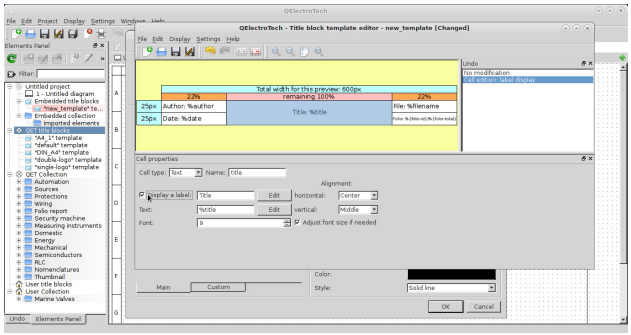
Total width and row height can be specified in 'pixels' while columns can be sized in absolute or in relative terms of '%' or 'pixels'



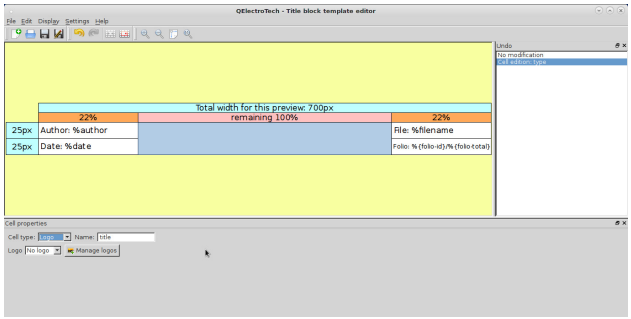
Double click row height cells



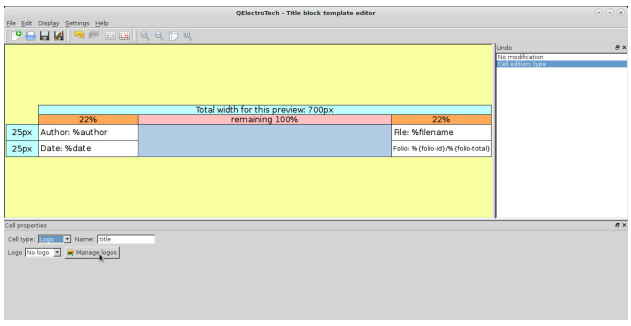
Single click a cell to display cell properties



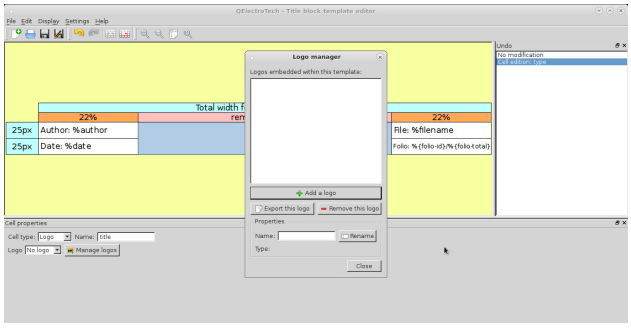
Selecting display a label adds the label to the cell



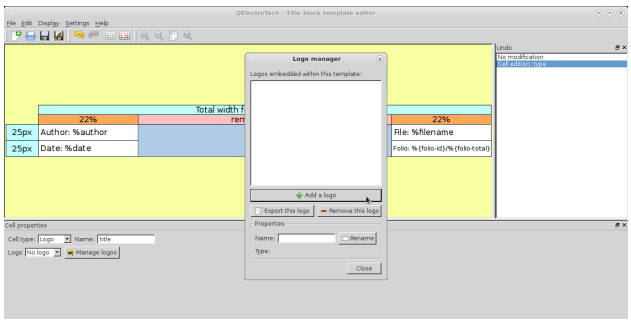
Changing cell type to 'logo'



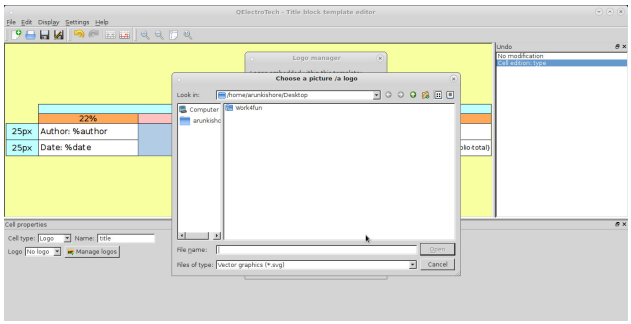
Click 'Manage logos' button to import logos



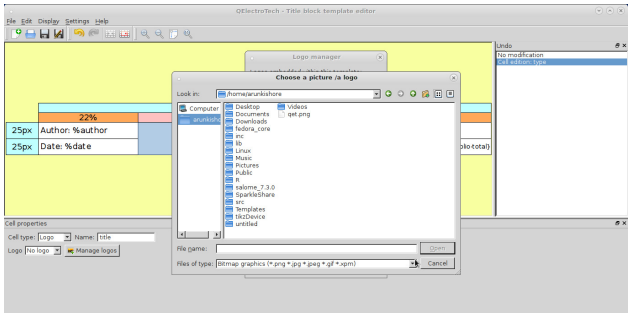
Logo manager



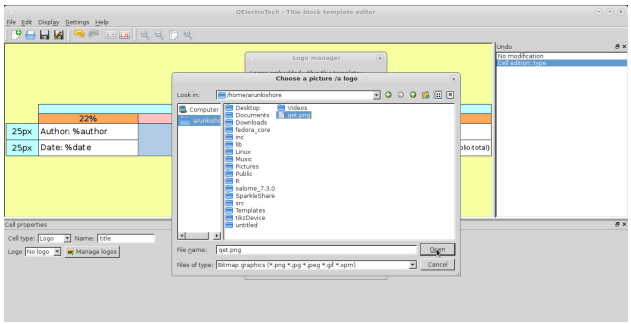
Click add a logo to search for logos



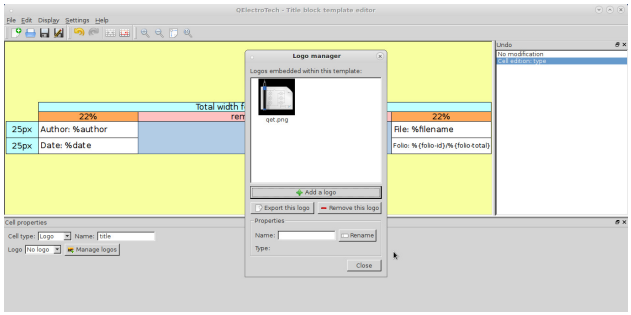
Choose a logo window opens with default type '*.svg'



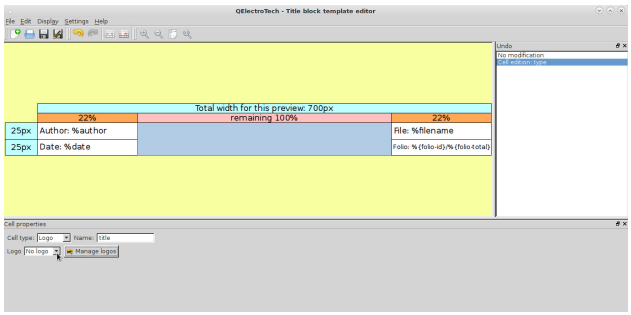
File types 'png', 'jpg','jpeg','gif' and 'xpm' are supported



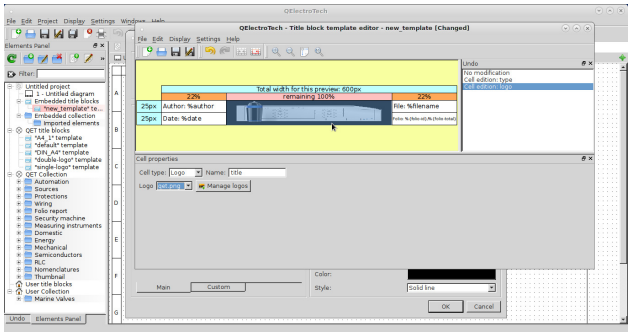
Choose a logo file and click 'OK'



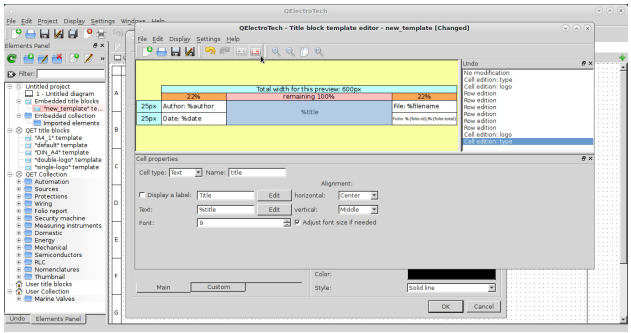
The logo is added, click close



Look for the added logo in the logo combo box

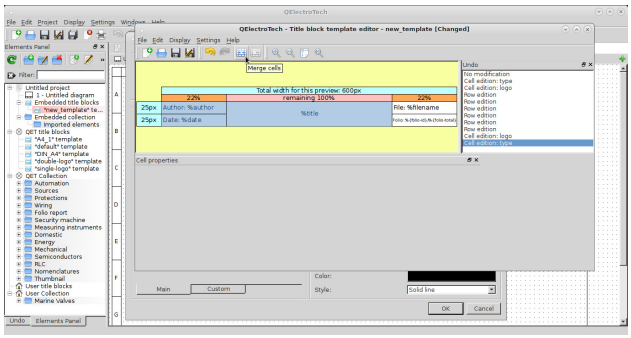


The logo is spread over the cell

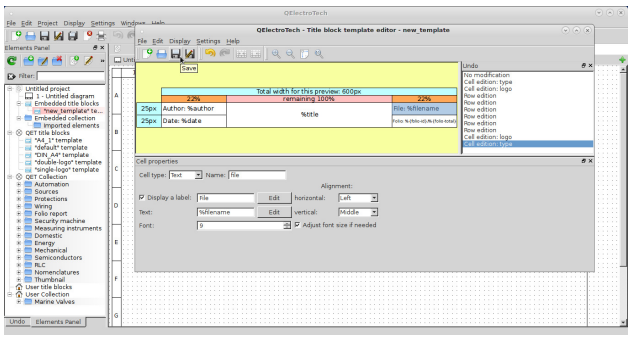


'Split cell' option activates for merged cells only

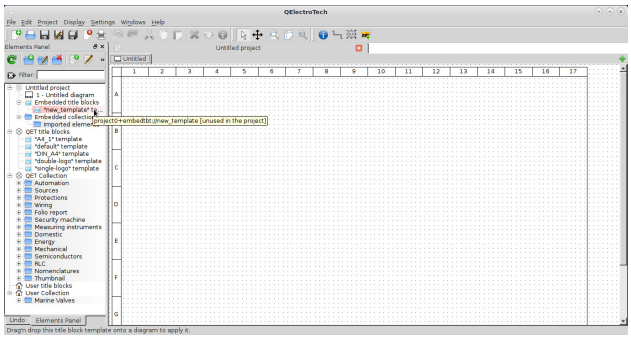
Note: Logo cell should be sized to match the logo aspect to prevent distortions. Image editor like GIMP may be used to measure the logo image aspect ratio. Logo cells can then be sized as illustrated earlier to maintain the aspect.



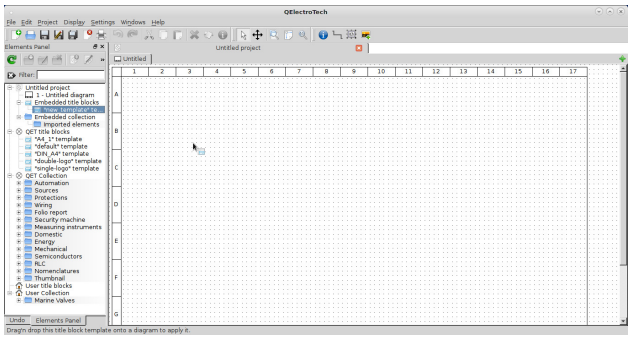
'Merge cells' option activates when multiple cells are selected with 'control'+left clicks



A template can be saved by clicking 'save' in tool bar

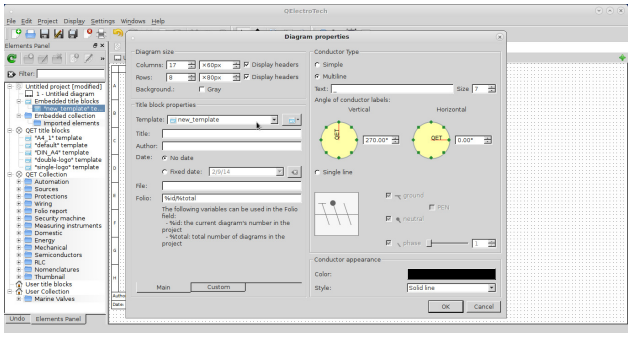


A saved template is visible under embedded template It is highlighted in light pink color

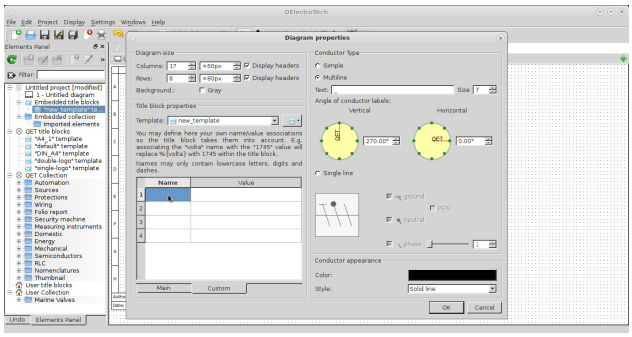


A saved template can be selected and drag dropped to apply to the diagram

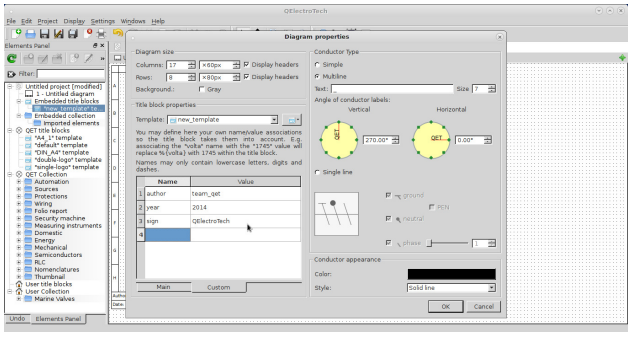
11.3 Custom tab



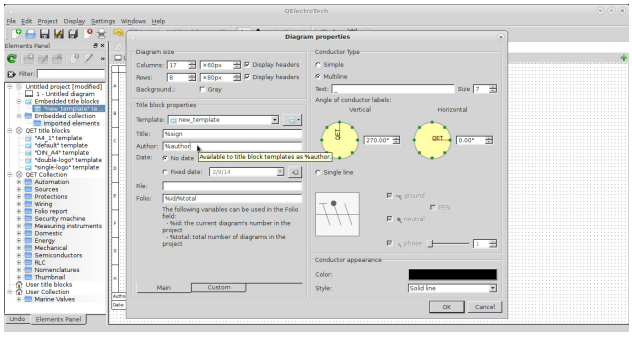
In the diagram properties select 'custom' tab



In the fields you can define a name and a value

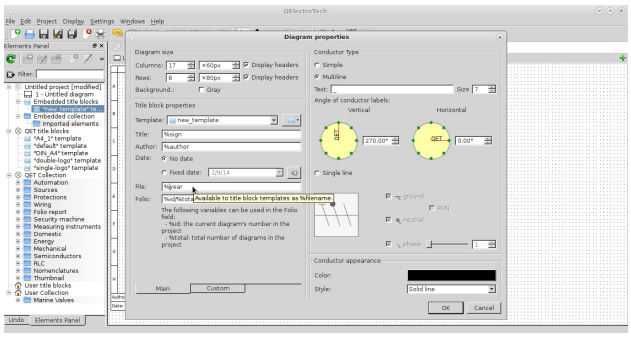


For example, some names and values are defined above

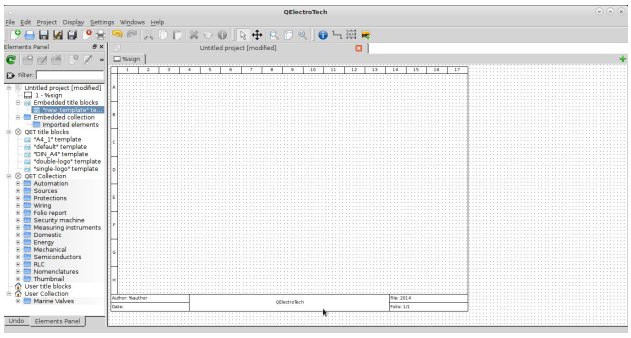


Switch back to 'main' tab and use the names as shown

'%' should be used before before the assigned names.

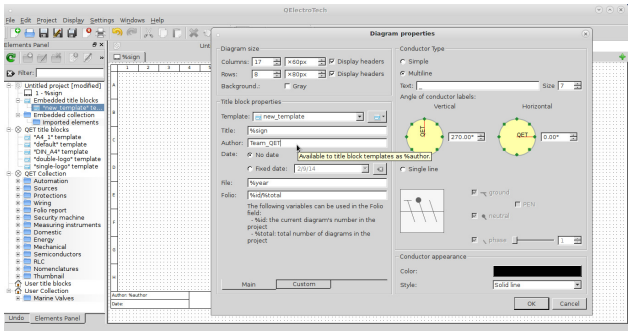


Assigned names are entered for all fields

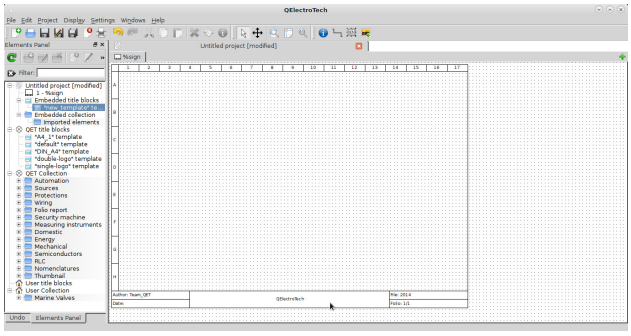


Watch the updated fields in the layout

'Author' field is not updated by its value, while other fields successfully show values for the names entered.



'Author' field is changed for proper display of name



Switch back to layout to watch the updated field

Note: Some names that are assigned a value may not update properly in the title block. Always verify for intended results. If some names are not updated by their values, edit the field to enter complete text from the title block editor. 'Custom tab' may not work for all variable names.

11.4 Working with the title block editor - (a tutorial)

In the following slides an example is illustrated, which will familiarize users with the title block editor. The following slides demonstrate how to imitate the title block from the referenced drawing in QElectroTech. The slides are in continuation to the slides illustrated under the section opening title block editor.

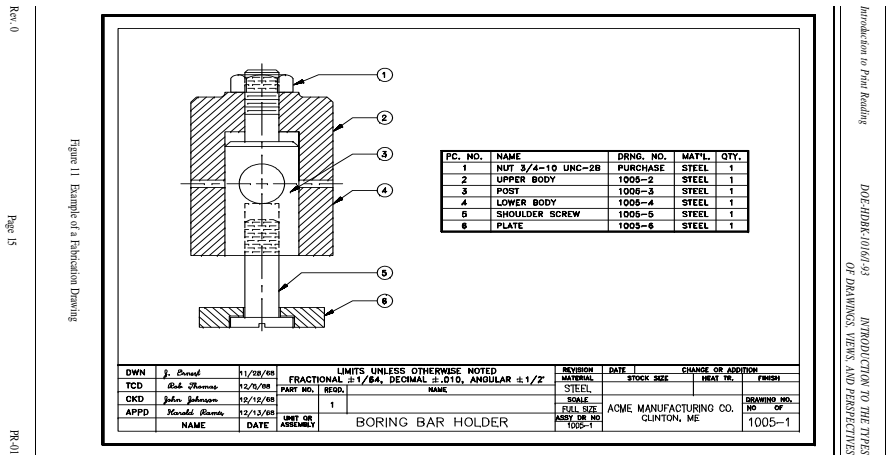
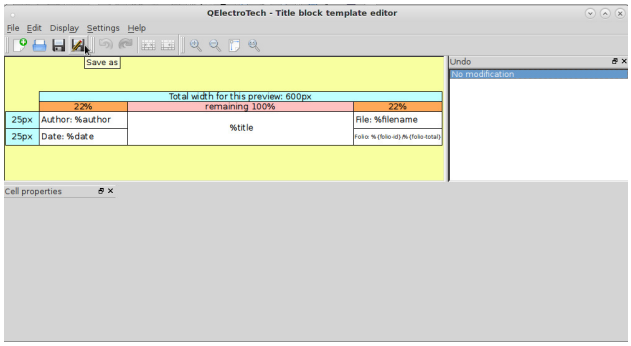
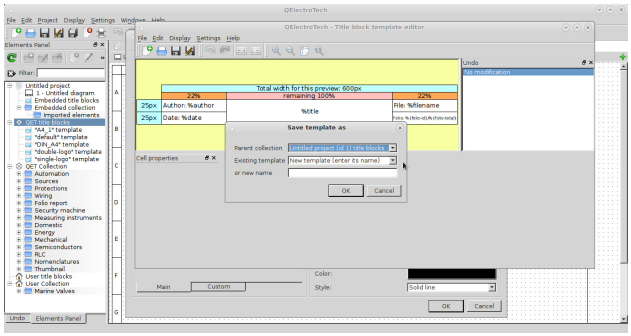


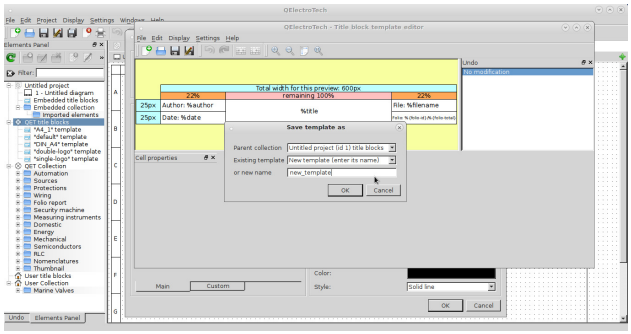
Figure 19: Reference drawing **(Source: DOE Handbook, Dept. of Energy, US)**



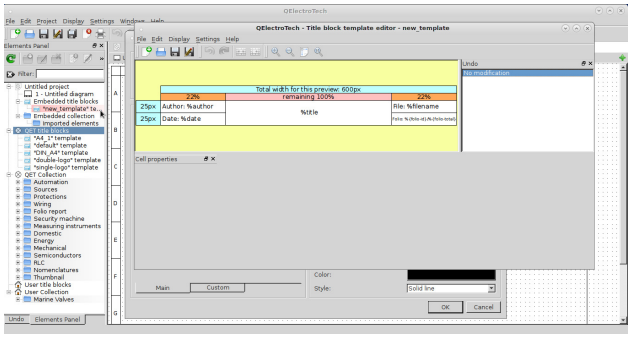
Click 'save as' button on tool bar



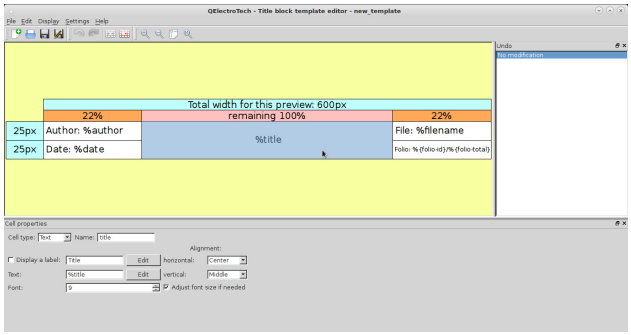
A window pops up to assign a name to the template



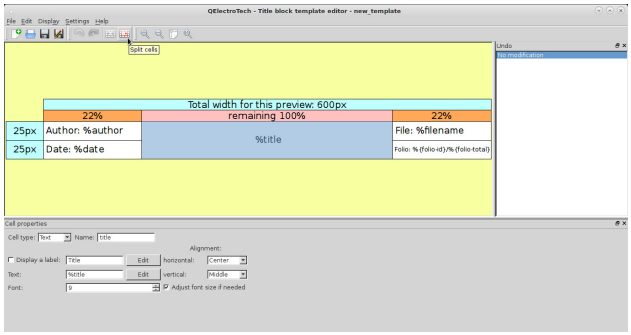
Give a name and click 'OK' to continue



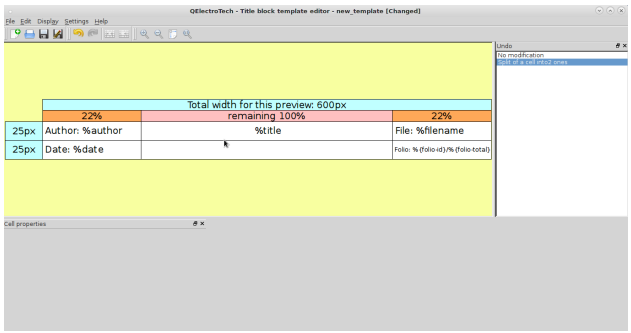
The template is now saved with the name



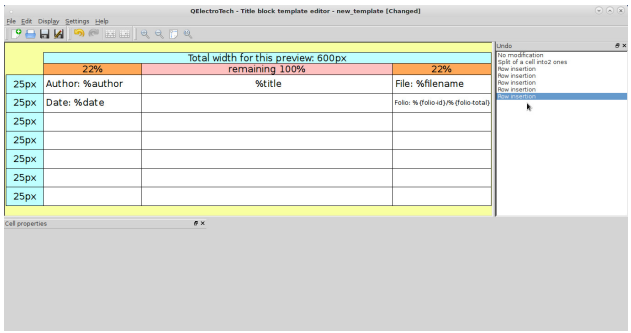
Select the title cell with left click



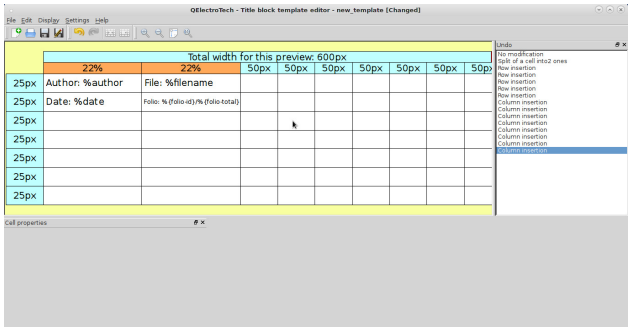
Click the split-cells button that is activated



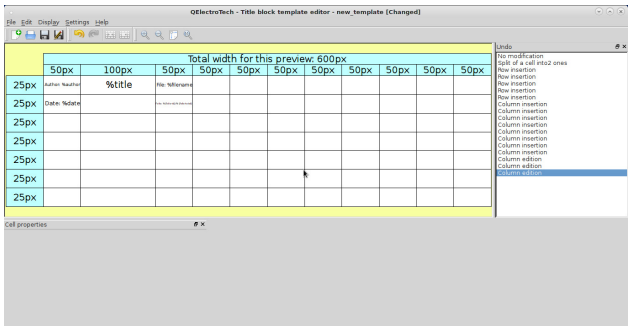
The 'split cell' reverts the earlier merged cells



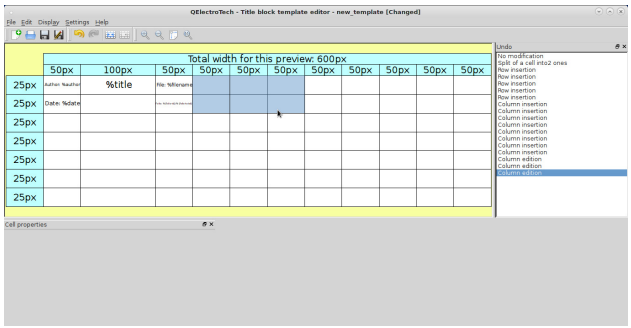
Add rows from 'Edit' option in tool bar to create 7 rows



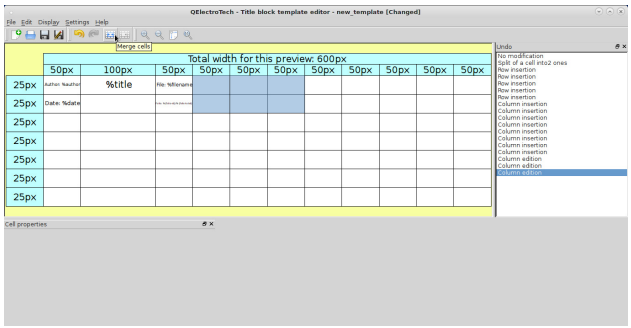
Add columns from ‘Edit’ to total 9 in number



Change column widths to absolute units as shown

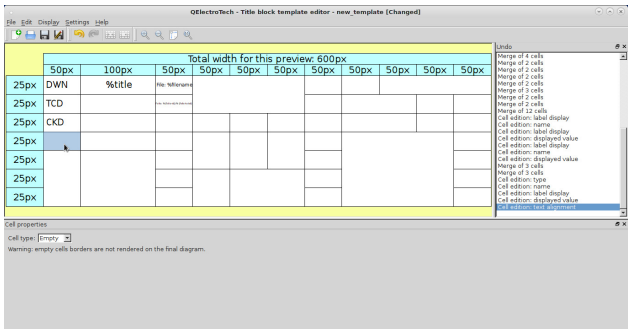


Select the cells using ‘control + left click’

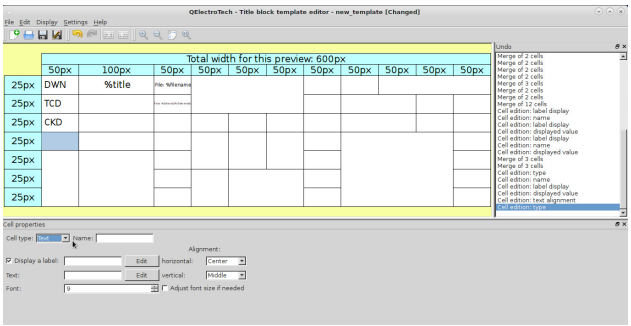


Click the ‘merge cells’ to merge selected cells

Follow similar actions to select a group of cells and merging them to arrive at the sketch illustrated below. This forms the skeleton structure for the intended title block.

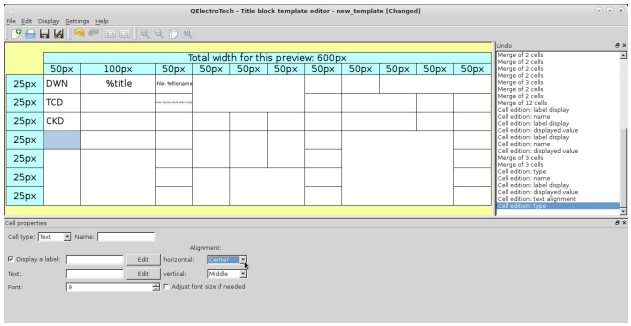


A skeleton structure of title block is achieved
Select the title cell with left click, the cell is empty

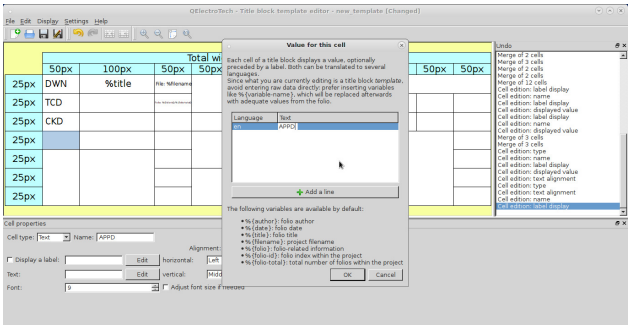


Change the cell type to text

Note: Empty cells will appear empty in the title block, no lines are drawn around it.



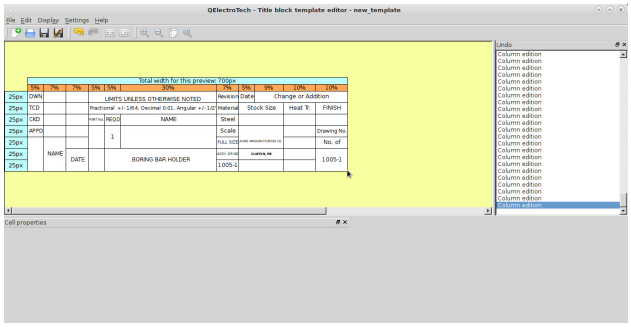
Set the alignment for the text in the cell
Uncheck ‘display label’ click ‘Edit’ to change text



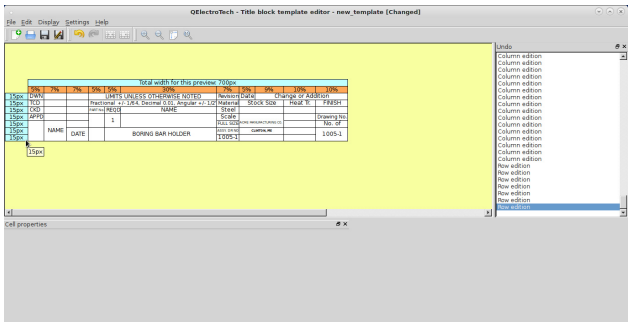
In the pop up window type ‘en’ for English under language
Double click the text field and type text to display

Text can be added to other fields as demonstrated - uncheck ‘display label’ and click the ‘Edit’ button next to the text field. A window to edit the text to be displayed pops up. A language code should be entered under language

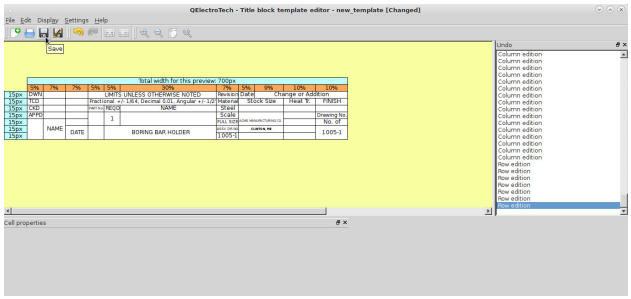
field. Type ‘en’ for English. Double click the text field to edit it, type text and click ‘OK’ to continue. Once all the text fields are filled up, the cells can be dimensioned suitably.



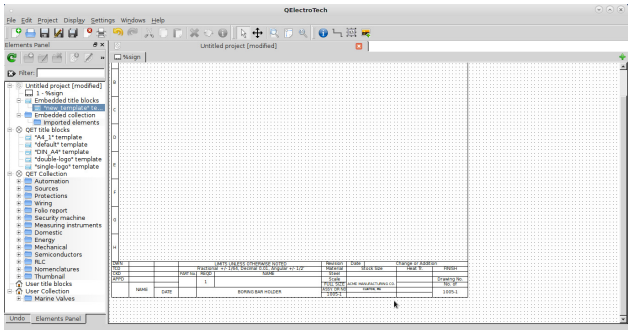
After text fields are updated the cells are dimensioned
Total width for preview is set to 700pixels
columns widths are set in relative terms as shown



Finally column heights are adjusted to 15pixels

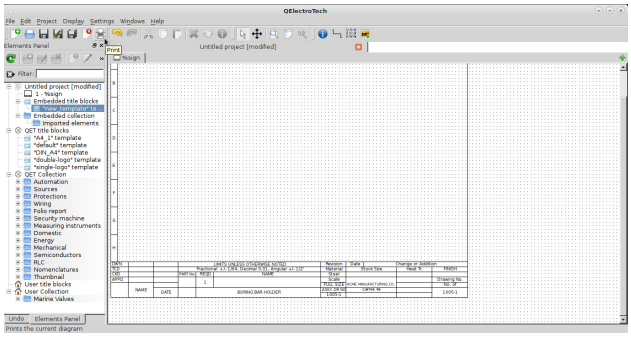


Click ‘Save’ button to save the title block
to the ‘new_template’

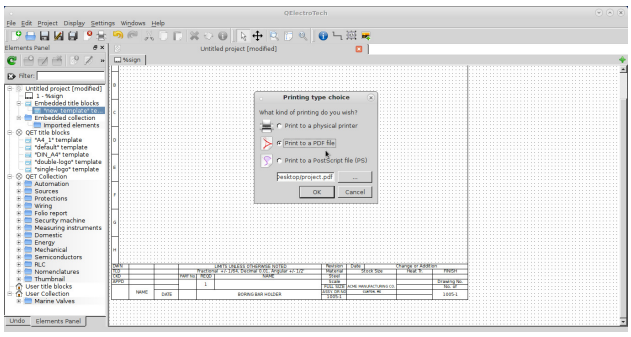


Watch the template in the QElectroTech window
The title block will be updated, reflecting changes

Note: Close the title block editor only after confirming that the changes are written to the template.



Click ‘print’ icon from the tool bar



A print wizard opens up, select ‘pdf’ out put format
Set file path and a name. Click ‘OK’ to continue

QElectroTech permits printing to a ‘physical printer’, ‘output to a pdf’ file and ‘output to a .ps’ file. Refer section 12.

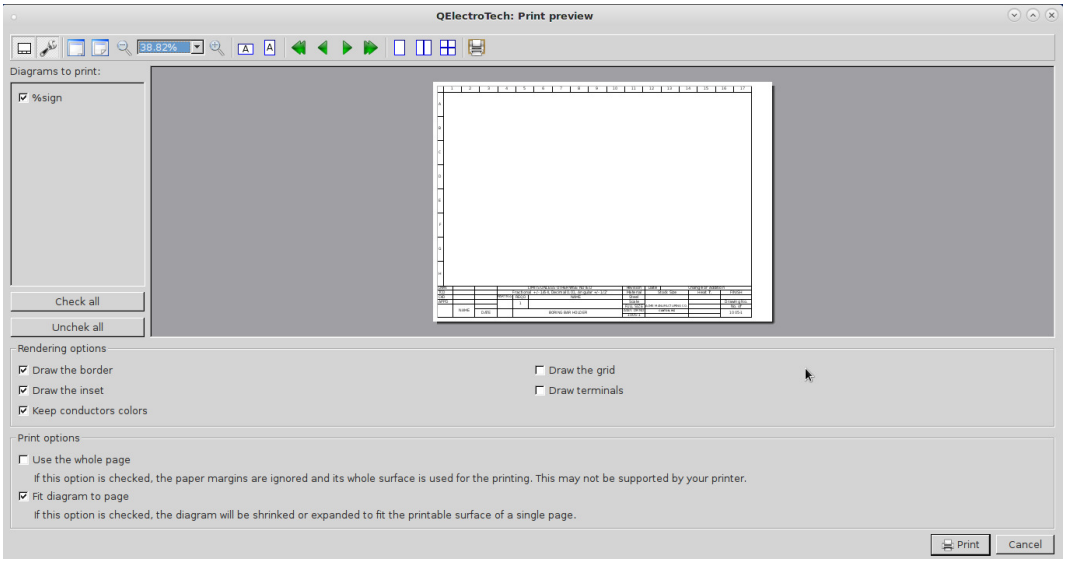


Figure 20: Print preview and properties window. Set properties and click 'print' to continue

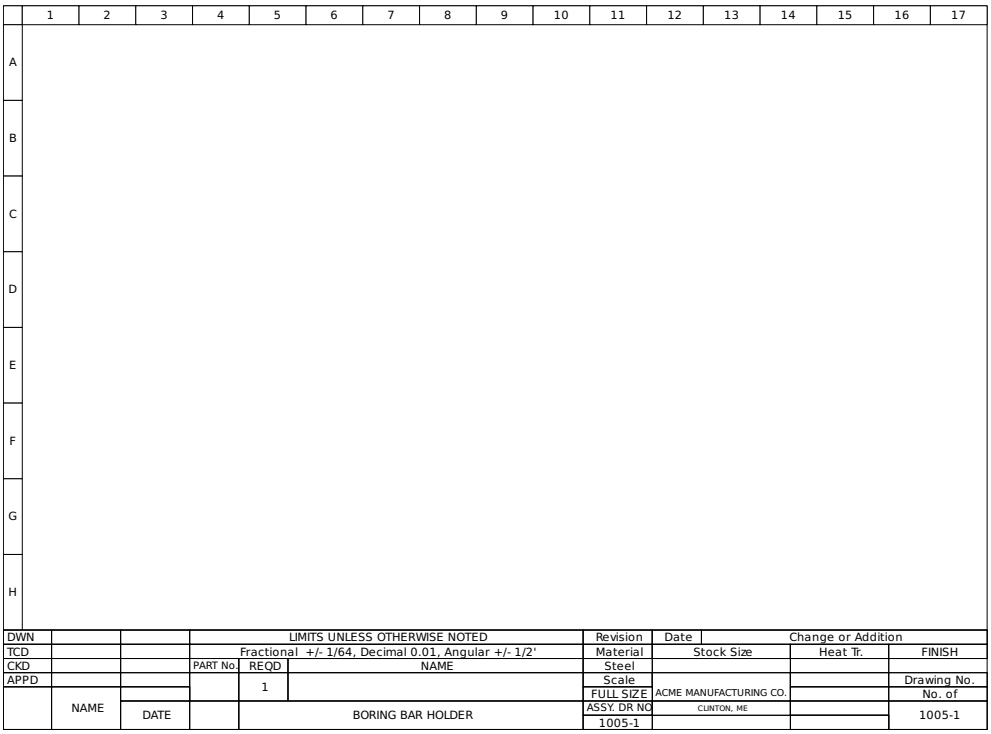


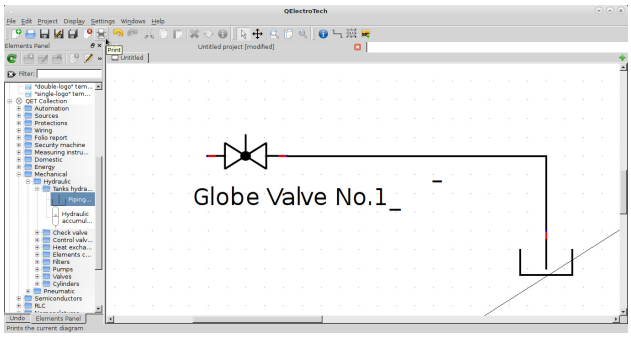
Figure 21: Finished print output of the template with intended title block

12 Printing, Exporting and Saving

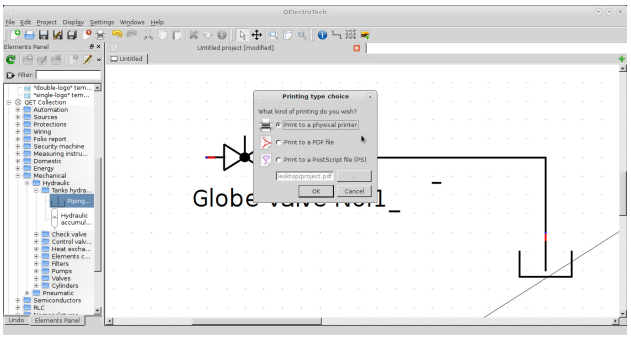
QElectroTech has excellent options to save, print or export your drawings created. The full features available are explained here in this section.

12.1 Printing

1. The completed drawings can be printed from a physical printer connected to the computer.
2. Drawings can be printed to a 'pdf' file to a selected directory.
3. Drawings can also be printed as '*.ps' (post script) files to a selected directory. Post script files are used for creating publishing quality graphics in LaTeX.



Select the 'print' button from the toolbar
Step 1



Print options can be set in the pop up window, click 'OK'.
Step 2

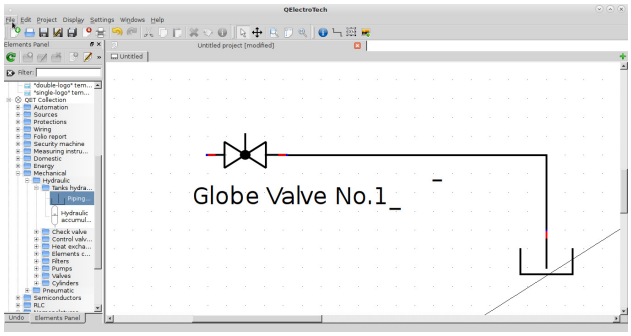
12.2 Saving

The completed drawings can be saved as '*.qet' project files in a directory. They can be reloaded for further changes or appended to a larger project. Click 'Save' or 'Save as' or 'Save the current diagram' from the main tool bar to save the drawing. File save wizard opens and facilitates locating the file in a directory in your computer.

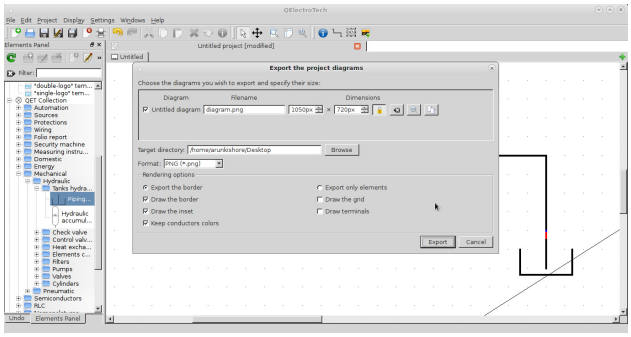
12.3 Exporting

1. QElectroTech can export drawings to 'PNG', 'JPEG', 'Bitmap', 'SVG' and 'DXF' formats.
2. The dimensions for export can be defined either with the same aspect ratio or even modifying aspect ratio.
3. A preview option is available to preview before exporting.
4. QElectroTech can also export the drawings to clipboard, which facilitates pasting the images in other applications.
5. QElectroTech also features a host of rendering options - a choice to select each or all of the options -
   (a) Export the border
   (b) Draw the border
   (c) Draw the inset
   (d) Keep conductors colors
   (e) Exporting only elements
   (f) Draw the grid
   (g) Draw the terminals

Document and graphics created under Creative Commons Attribution 3.0 License



Select the 'File' button from the main menu bar
Step 1



Select export from the drop down window
Step 2

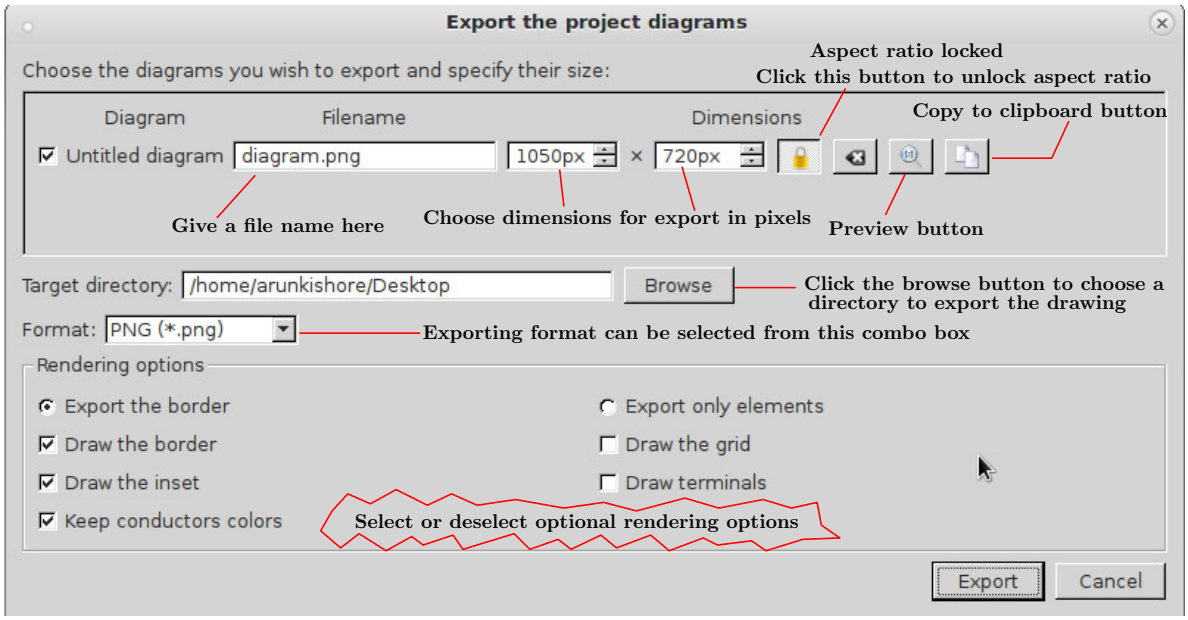
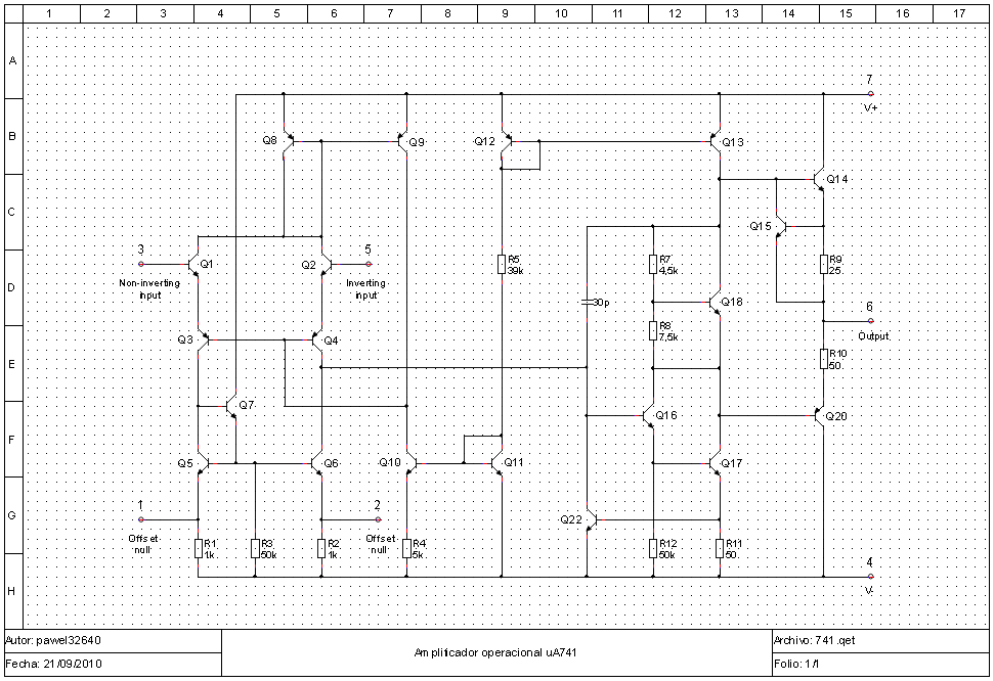


Figure 22: Export wizard in QElectroTech

Step 3: Choose your export options and click 'Export'

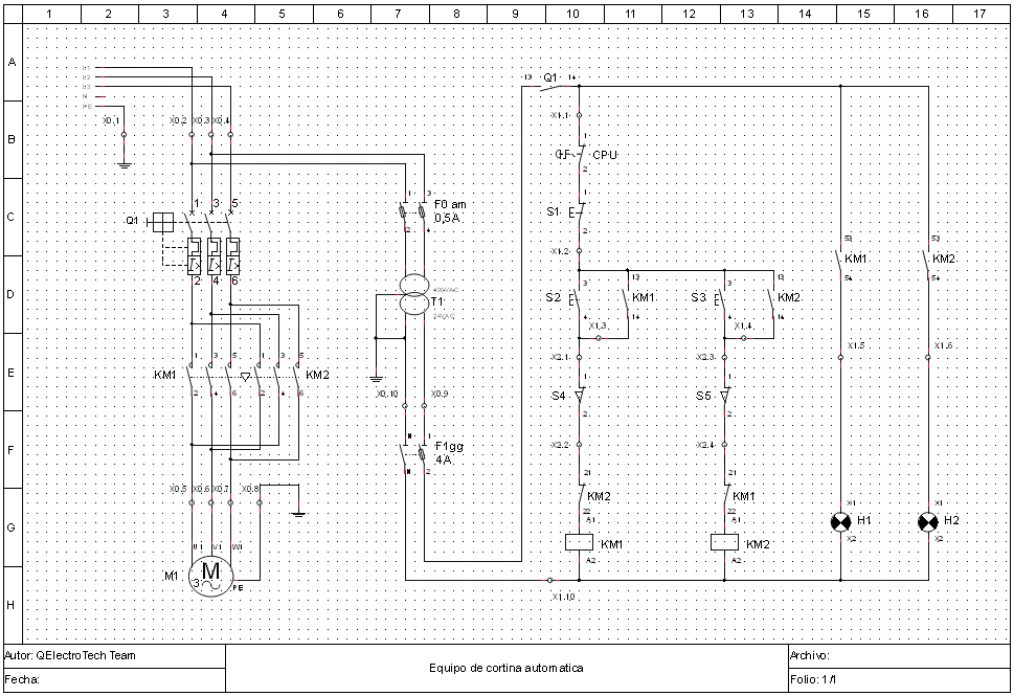
13 Some sample drawings created in QElectroTech

741.png (PNG Image, 969 × 666 pixels) - Scaled ... http://download.tuxfamily.org/qet/manuals/es/741.png



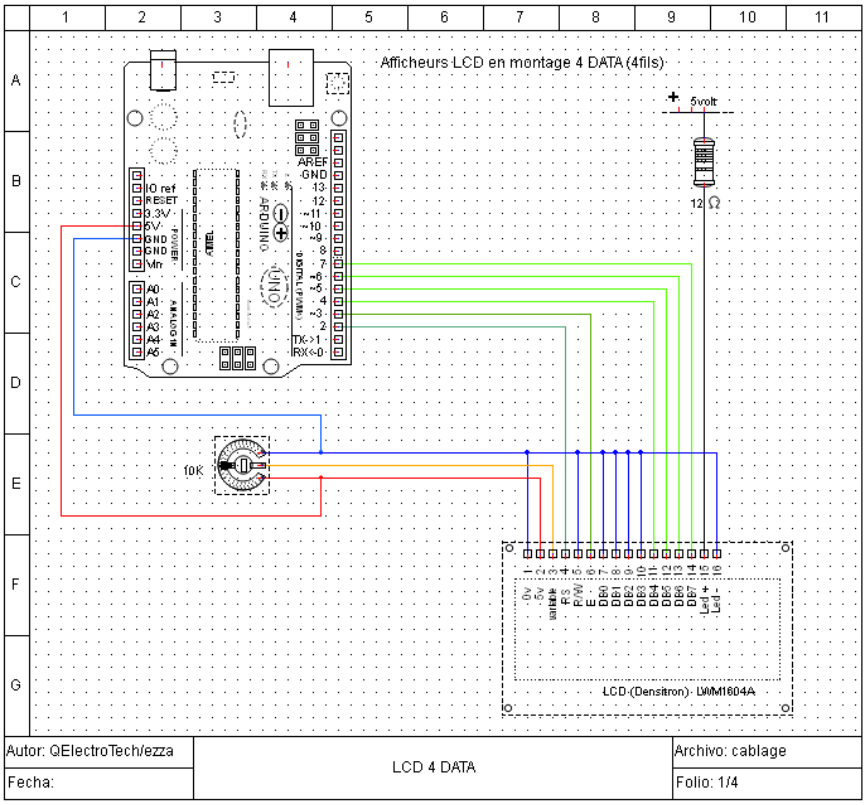
Document and graphics created under Creative Commons Attribution 3.0 License

schema_indus.png (PNG Image, 967 × 663 pixels)... http://download.tuxfamily.org/qet/manuals/es/sc...



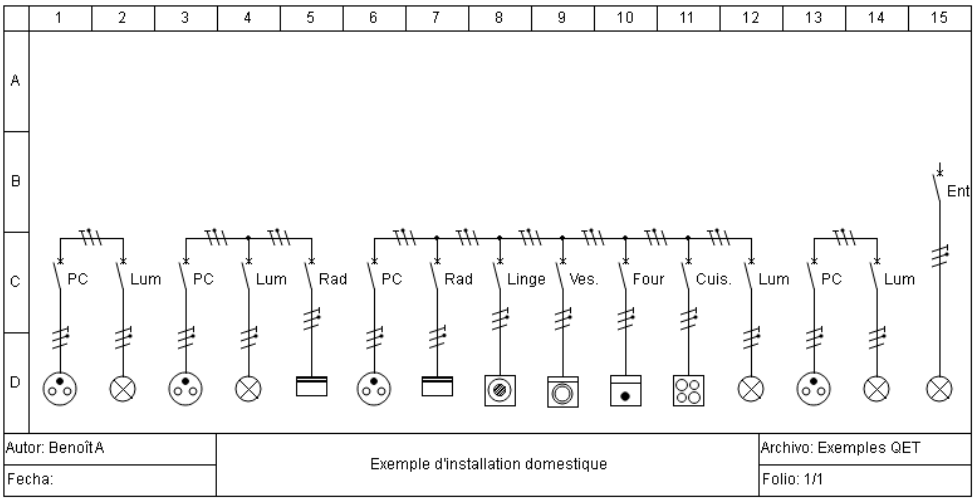
Document and graphics created under Creative Commons Attribution 3.0 License

arduino-lcd.png (PNG Image, 686 × 637 pixels) - ... http://download.tuxfamily.org/qet/manuals/es/ard...



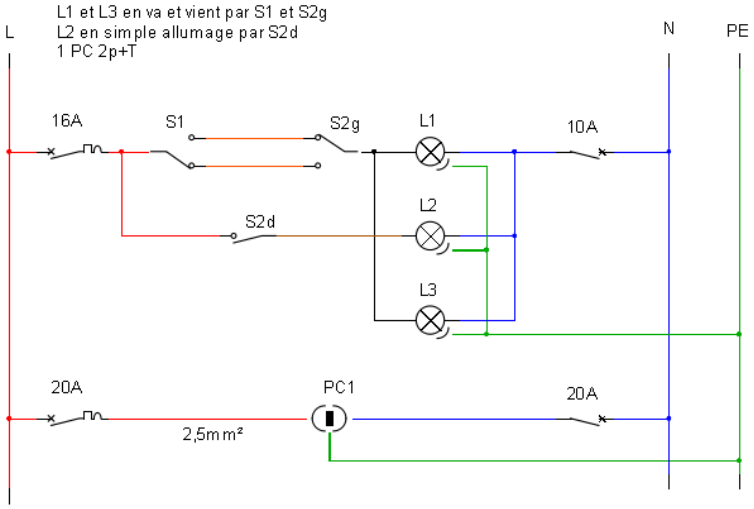
Document and graphics created under Creative Commons Attribution 3.0 License

exemple_dinstallation_domestique.png (PNG Imag... http://download.tuxfamily.org/qet/manuals/es/exe...

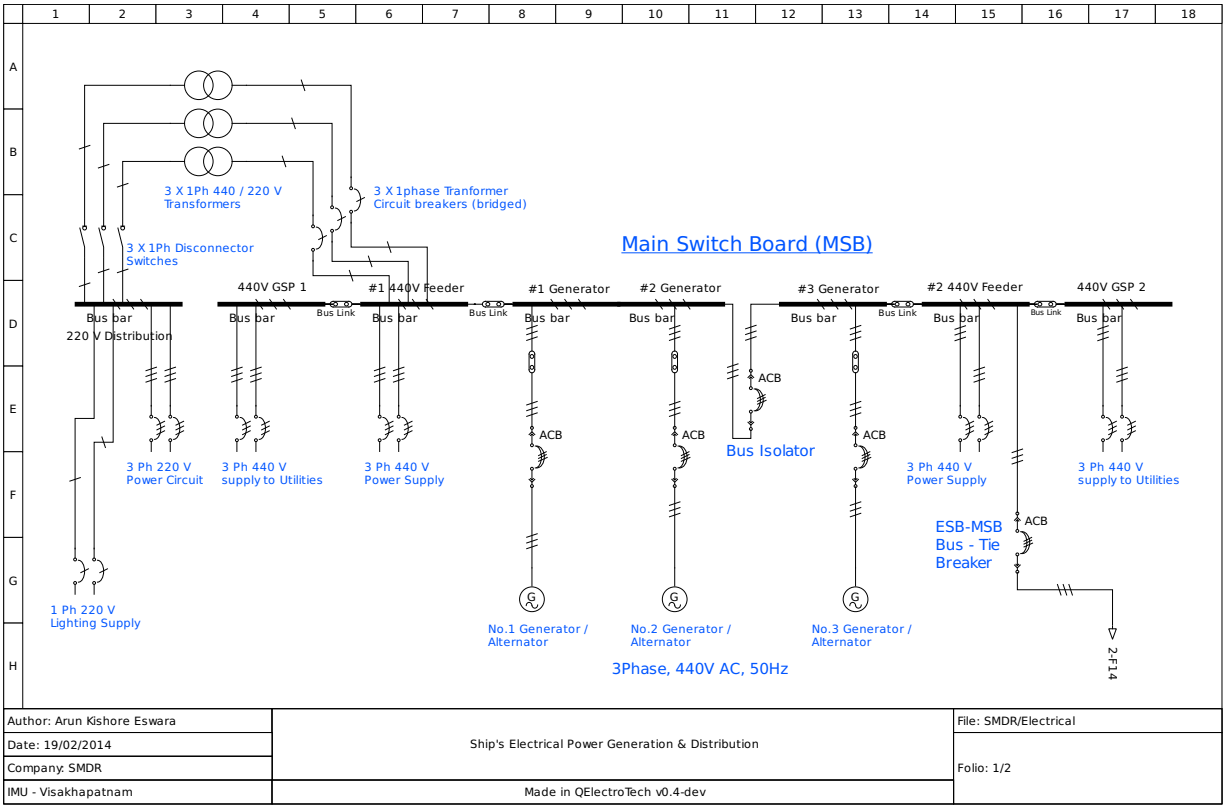


1 of 1 02/01/2014 09:28 PM

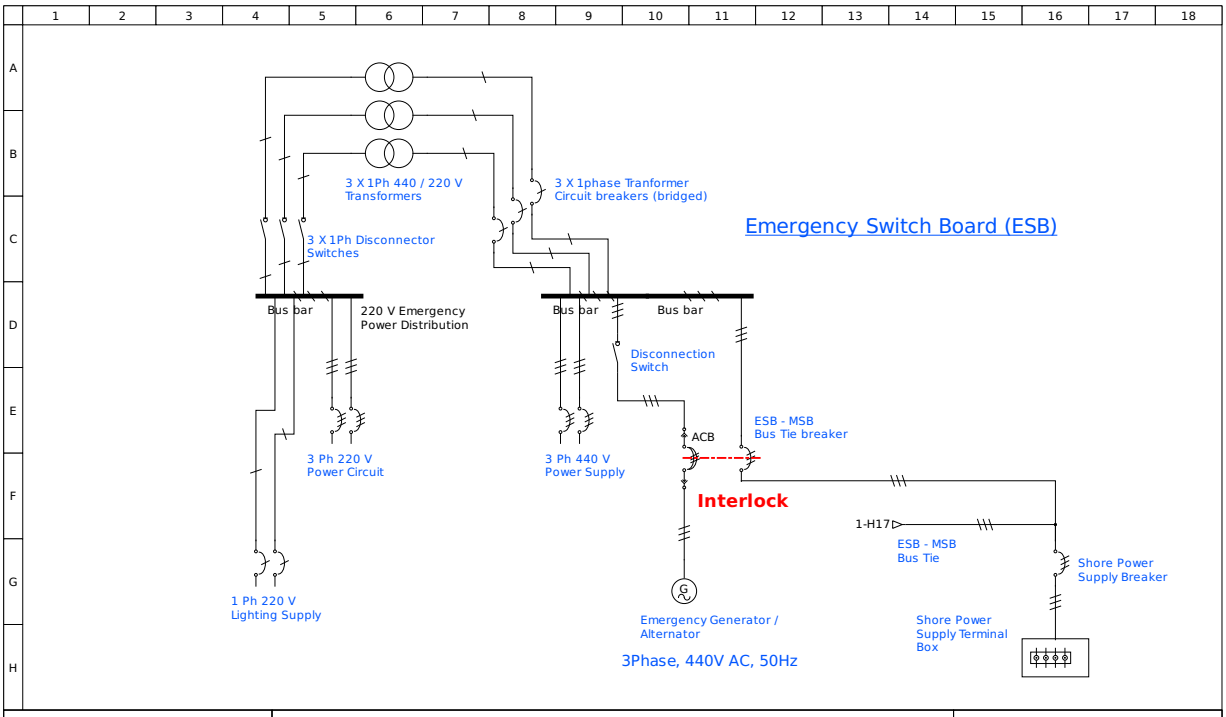
eclairage_va_et_vient.png (PNG Image, 630 × 439... http://download.tuxfamily.org/qet/manuals/es/ecla...



1 of 1 02/01/2014 09:28 PM



Author: Arun Kishore Eswara	Ship's Electrical Power Generation & Distribution	File: SMDR/Electrical
Date: 19/02/2014		Folio: 1/2
Company: SMDR		
IMU - Visakhapatnam	Made in QElectroTech v0.4-dev	



Author: Arun Kishore Eswara	Ship's Emergency Power Generation & Distribution	File: SMDR / Electrical
Date: 19/02/2014		Folio: 2/2
Company: SMDR		
IMU - Visakhapatnam	Made in QElectroTech v0.4-dev	