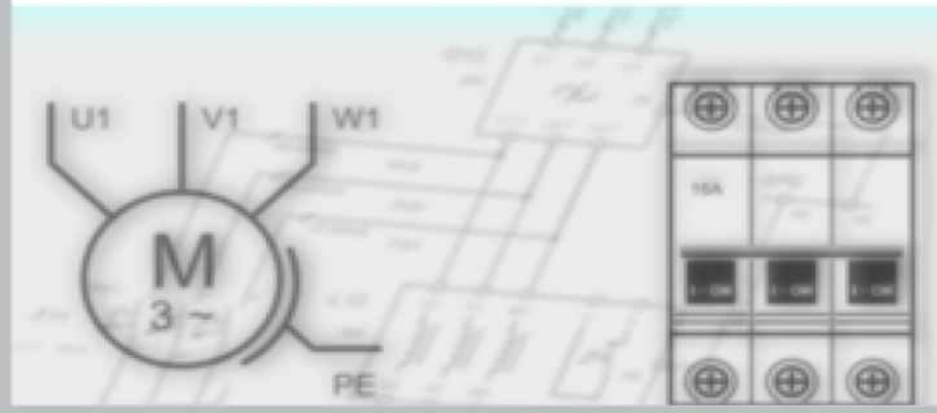
Master drawing diagrams with QElectroTech

My user guide to Master drawing diagrams with QElectroTech

ADIL RAJI

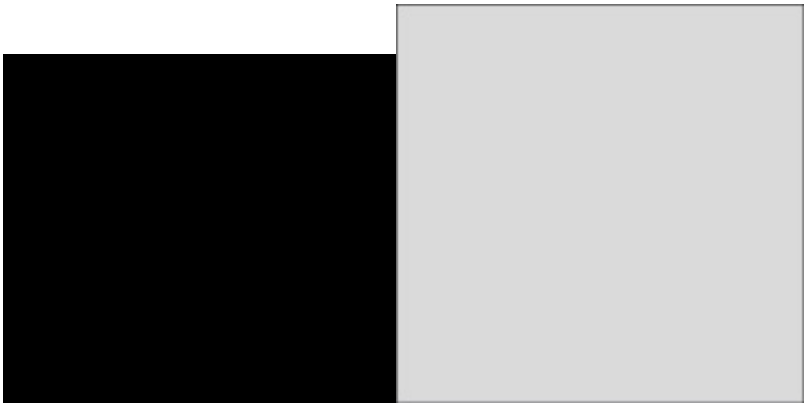
QElectroTech

free electrical diagram v0.6



- International
- Multi OS
- Open source
- Community forum

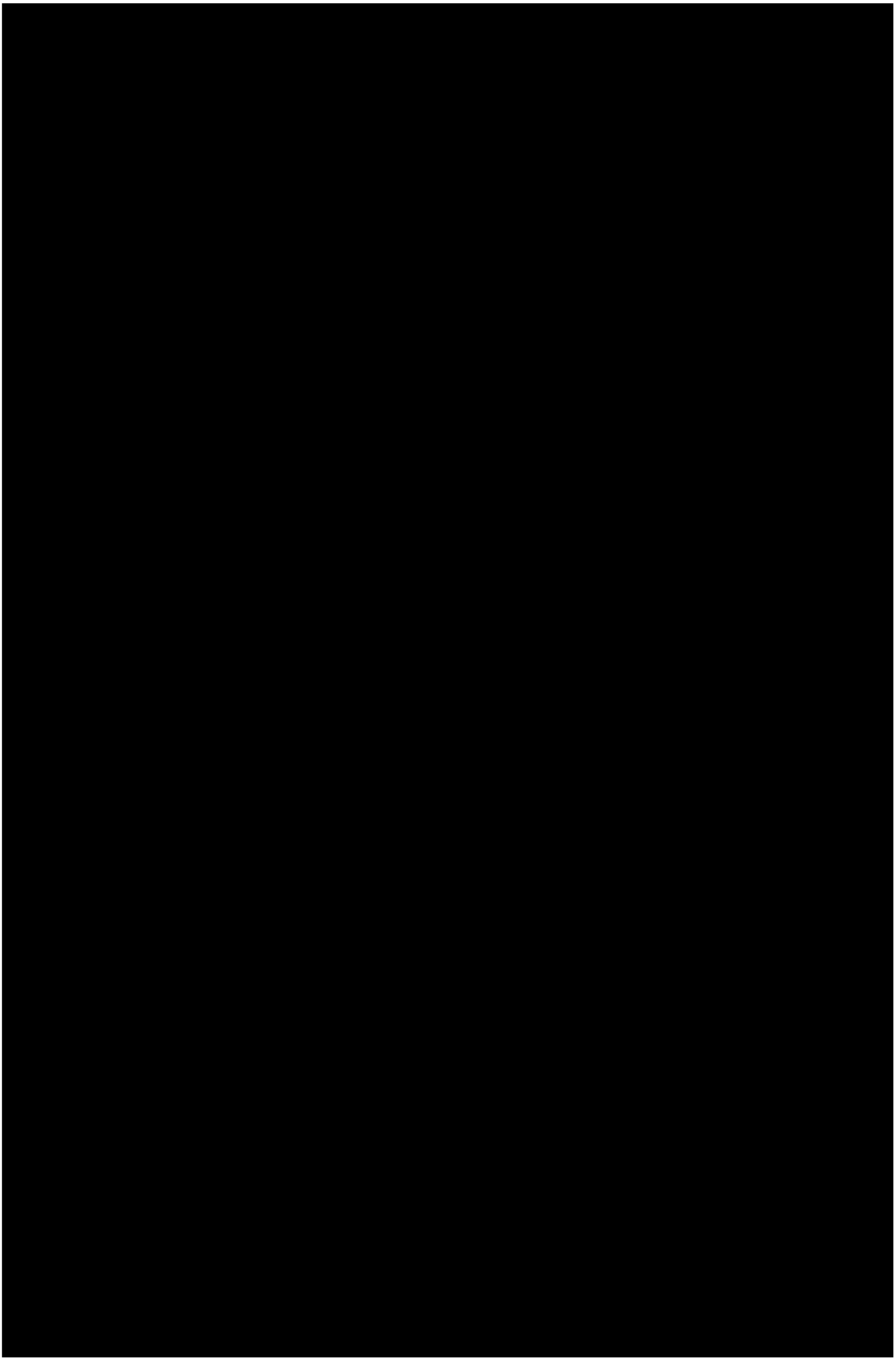
NOV 2018



My user guide to:

Master drawing diagrams with QElectroTech

By Mr.Adil RAJI



Summary:

Chapter One I- Introduction

1 Introduction:

QElectroTech is used to draw electrical and / or electronic diagrams;
QElectroTech is not used to simulate the operation of diagrams;
QElectroTech is for those who wish to build an electrical file (images, paper,...)
QElectroTech is not for those who want to test the operation, study the characteristics of circuits

or other studies;
This book is for those who wish to learn how to use QElectroTech to draw industrial electrical diagrams.

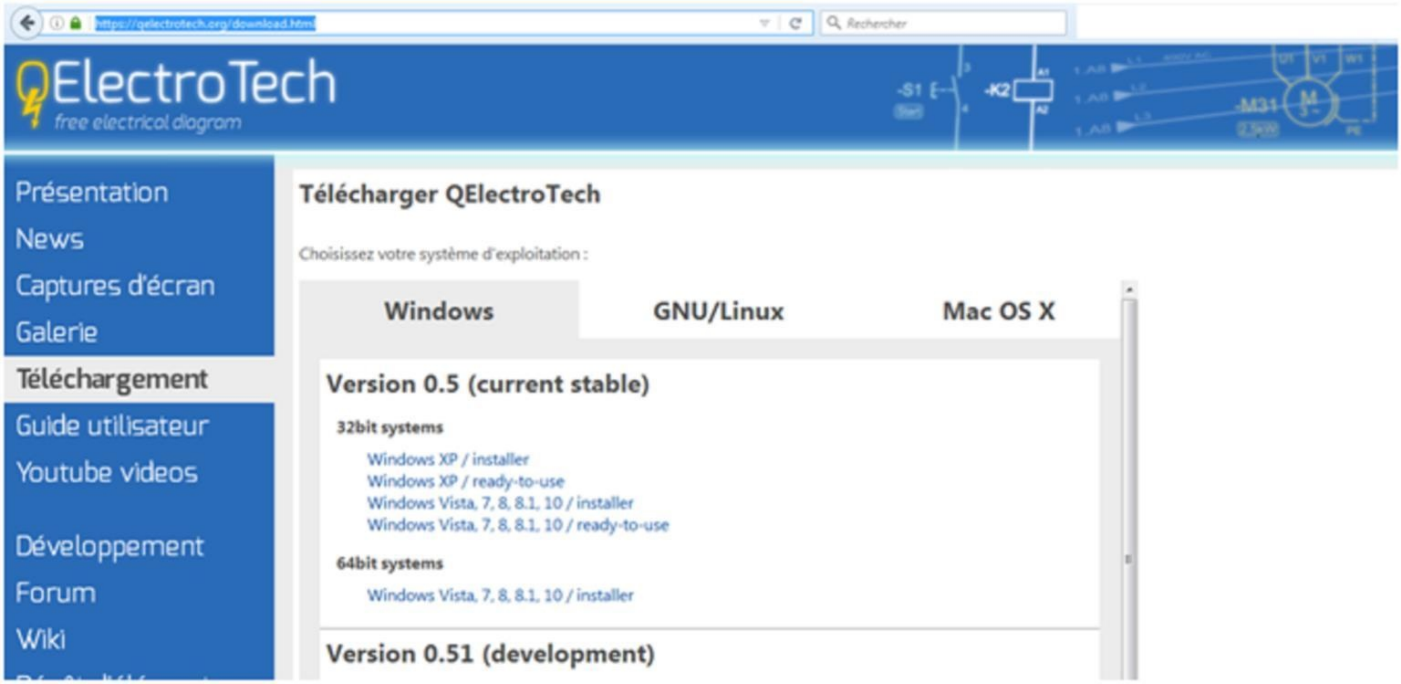
1.1 Objectives:

This book aims to introduce you to QElectroTech software.
What I propose as methods in this book is drawn from my experience, and is not the only way to go.
I have tried to present in this book the necessary functions to the realization of the standard industrial electrical diagrams.

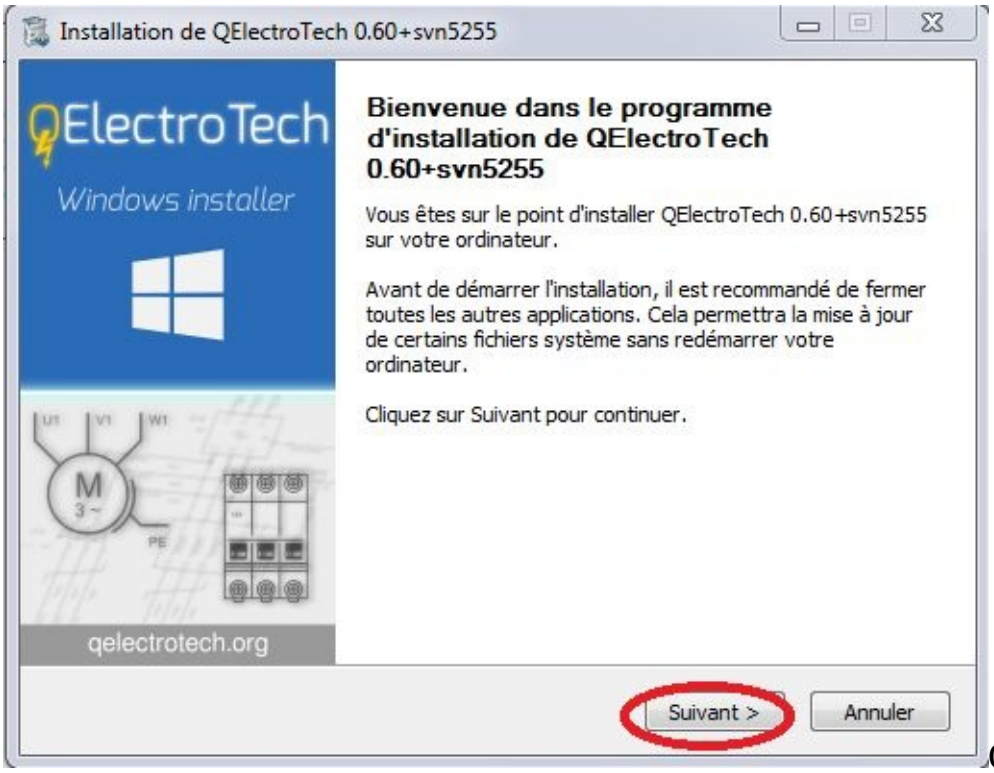
You have to be careful, to get the most out of this book follow exactly the steps described to master my way of doing things and then you can explore the rest of QElectroTech's features and build your own way of working.

1.2 Download & installation:

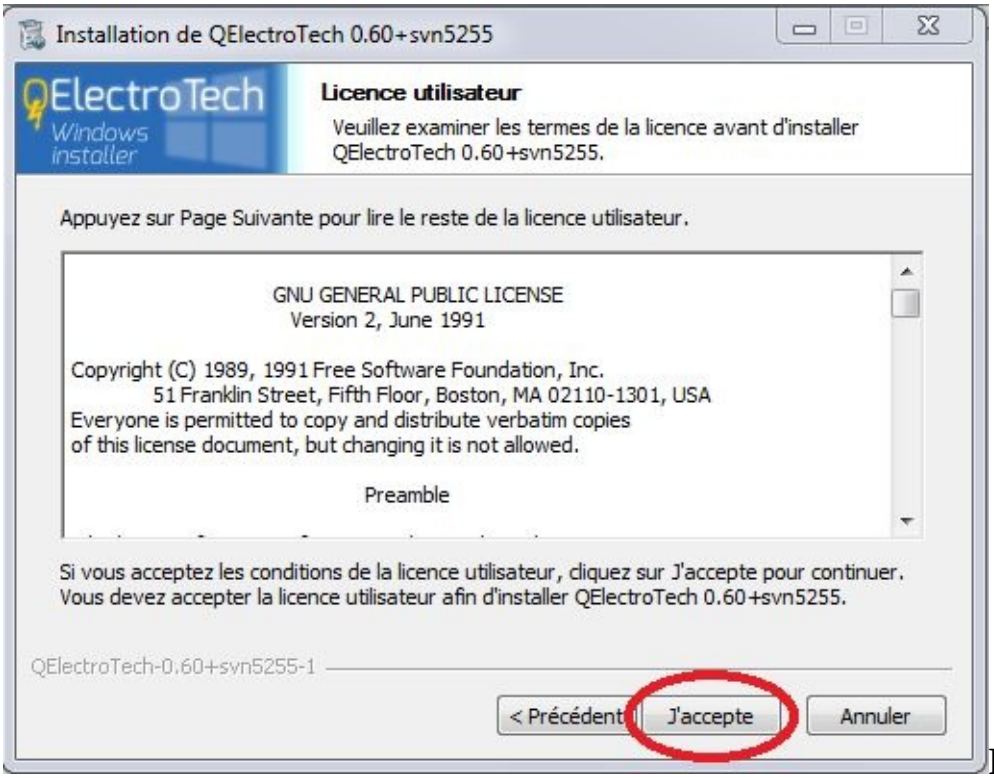
To install the software, here is the link to the QElectroTech download page:
https://qelectrotech.org/download.html



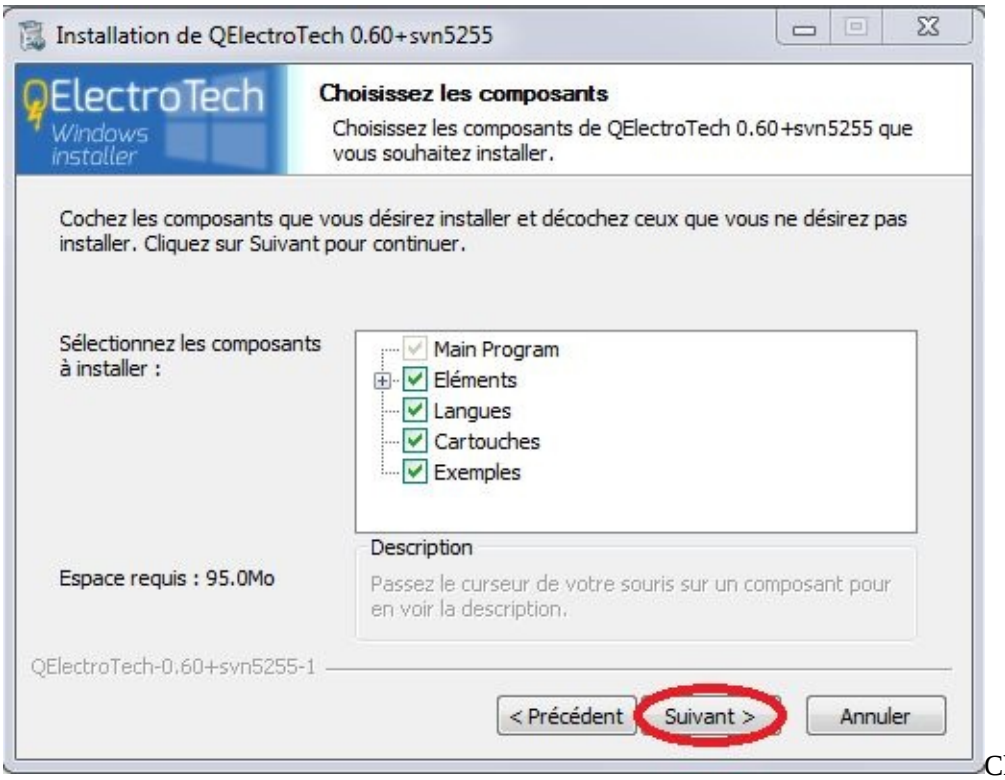
Select your operating system to start the download.
At the end of the download start the execution and follow the installation steps.



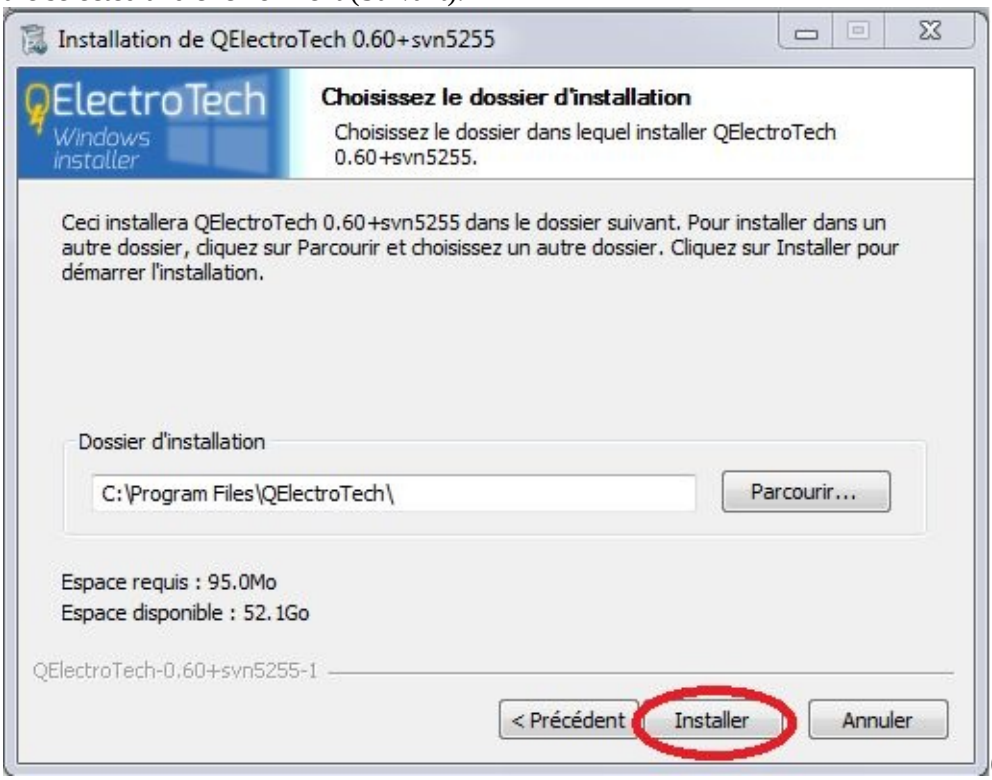
Choose next (Suivant).



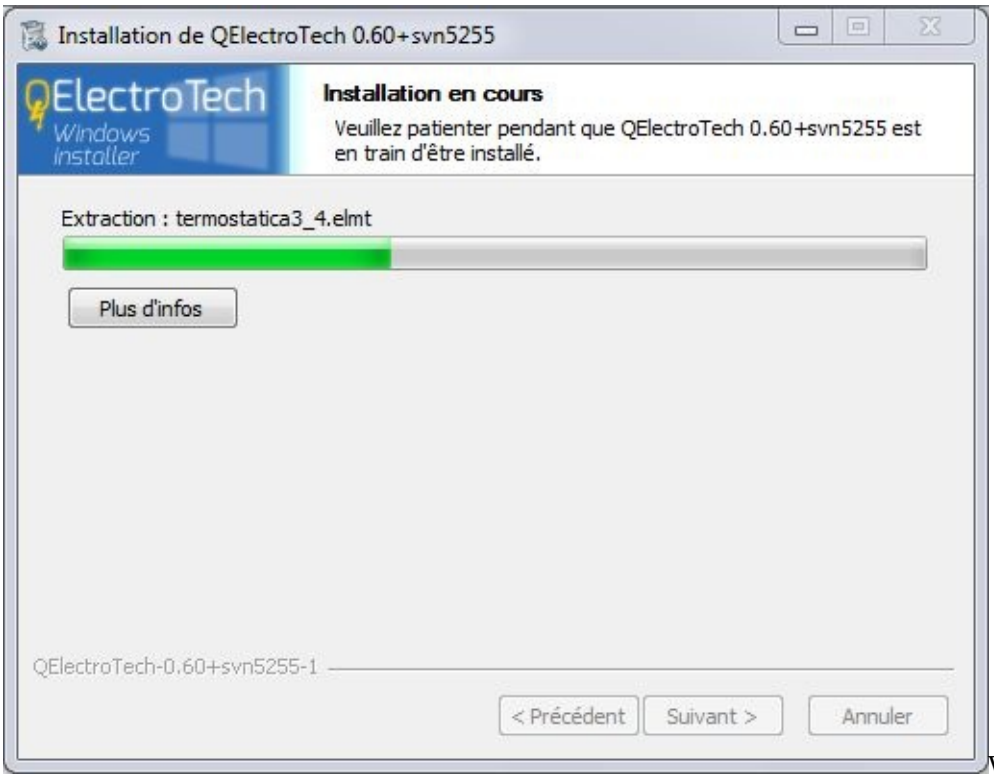
Read and accept the terms of the user license(J’accepte).



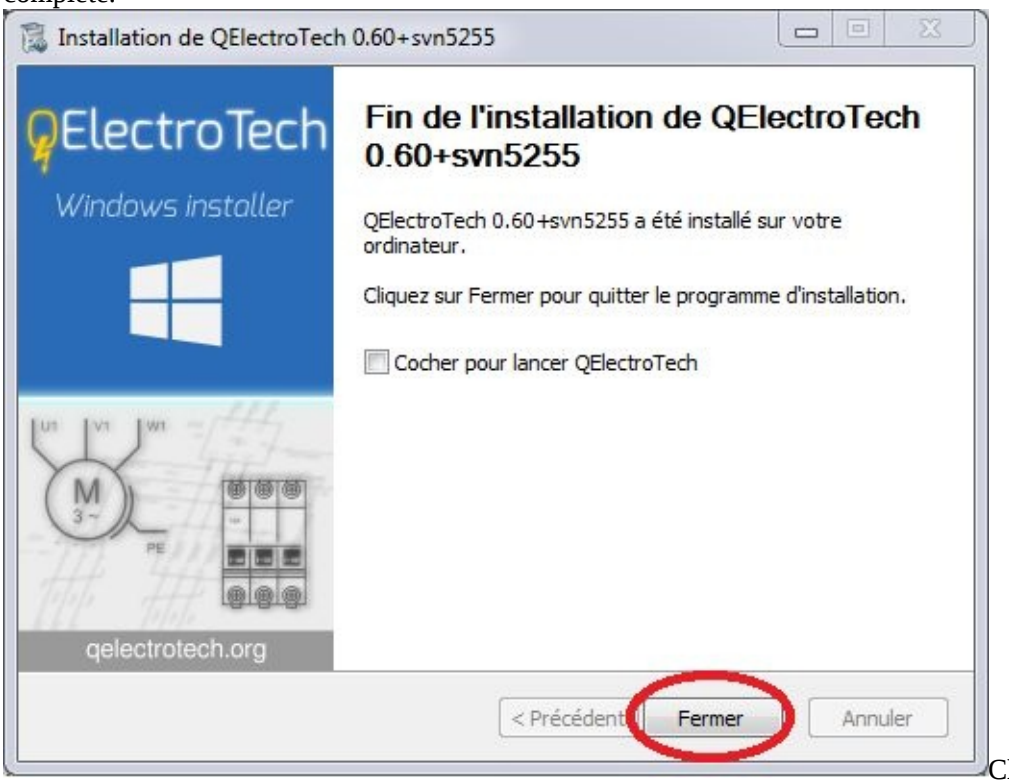
Check if all program components are selected and click on next (Suivant).



Click to Install (Installer).



Wait for the installation to be complete.



Click on close (Fermer).

And here you have just completed the installation, and you need to set the language.

Chapter two II- Presentation of QElectroTech

2 Presentation of QElectroTech:

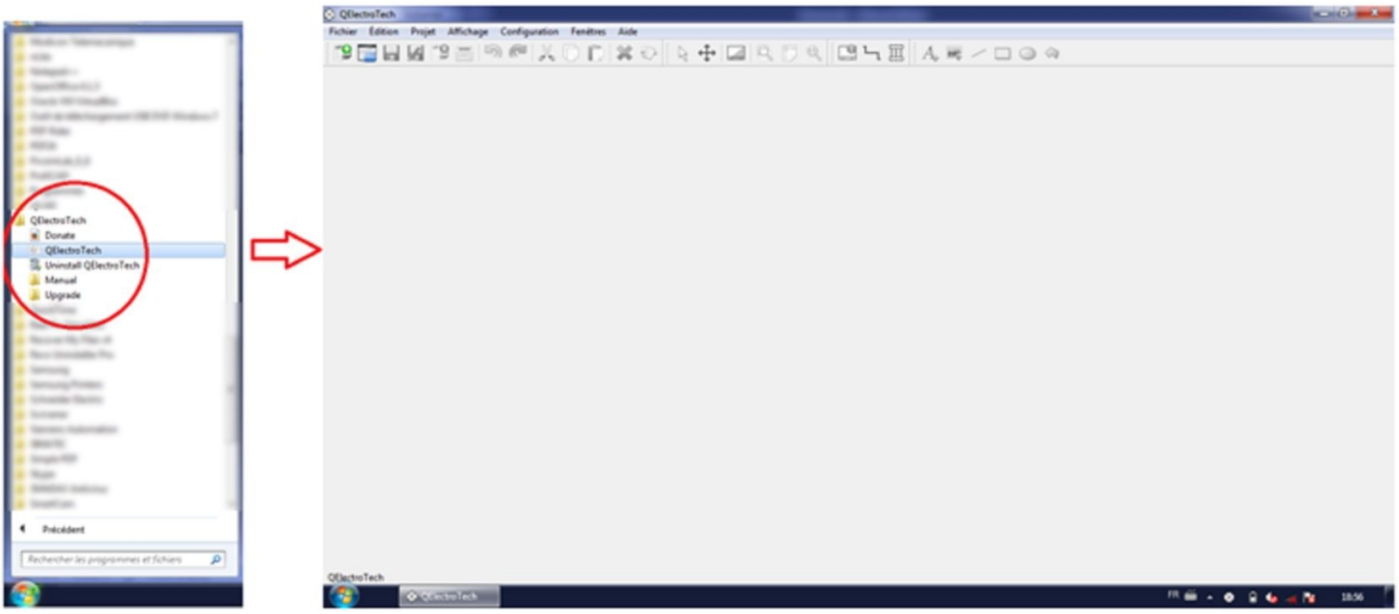
QElectroTech: Designed for electricians, and is free software for drawing electrical diagrams with the ability to create projects, elements and collections in several categories; it also helps in the creation of electronic diagrams.

Version: QElectroTech is currently available in version 0.6 and is under license GNU / GPL. **Operating System**: QElectroTech is currently available for Windows and Linux. **Simulation**: For the moment QElectroTech does not allow to carry out simulations. Level of difficulty: QElectroTech is simple and easy to use software with simple drag and drop

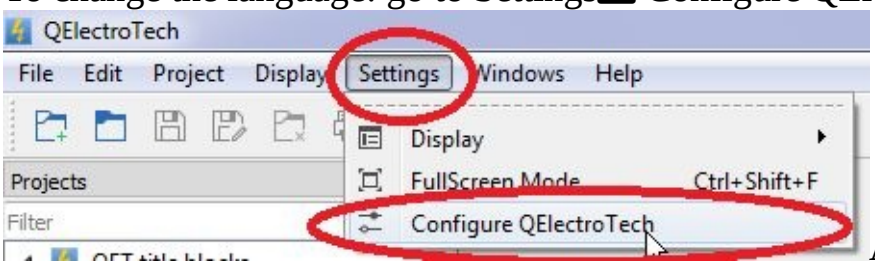
you can do most manipulations to make the drawings and with these options of page jump you can easily navigate and follow the course of the circuit.
Main advantage: QElectroTech is much easier than other paid software like. **Main disadvantage**: for those who want to make simulation of their electrical design and drawings for the moment they cannot do it with QElectroTech.

2.1 Launching and creating your first folio:

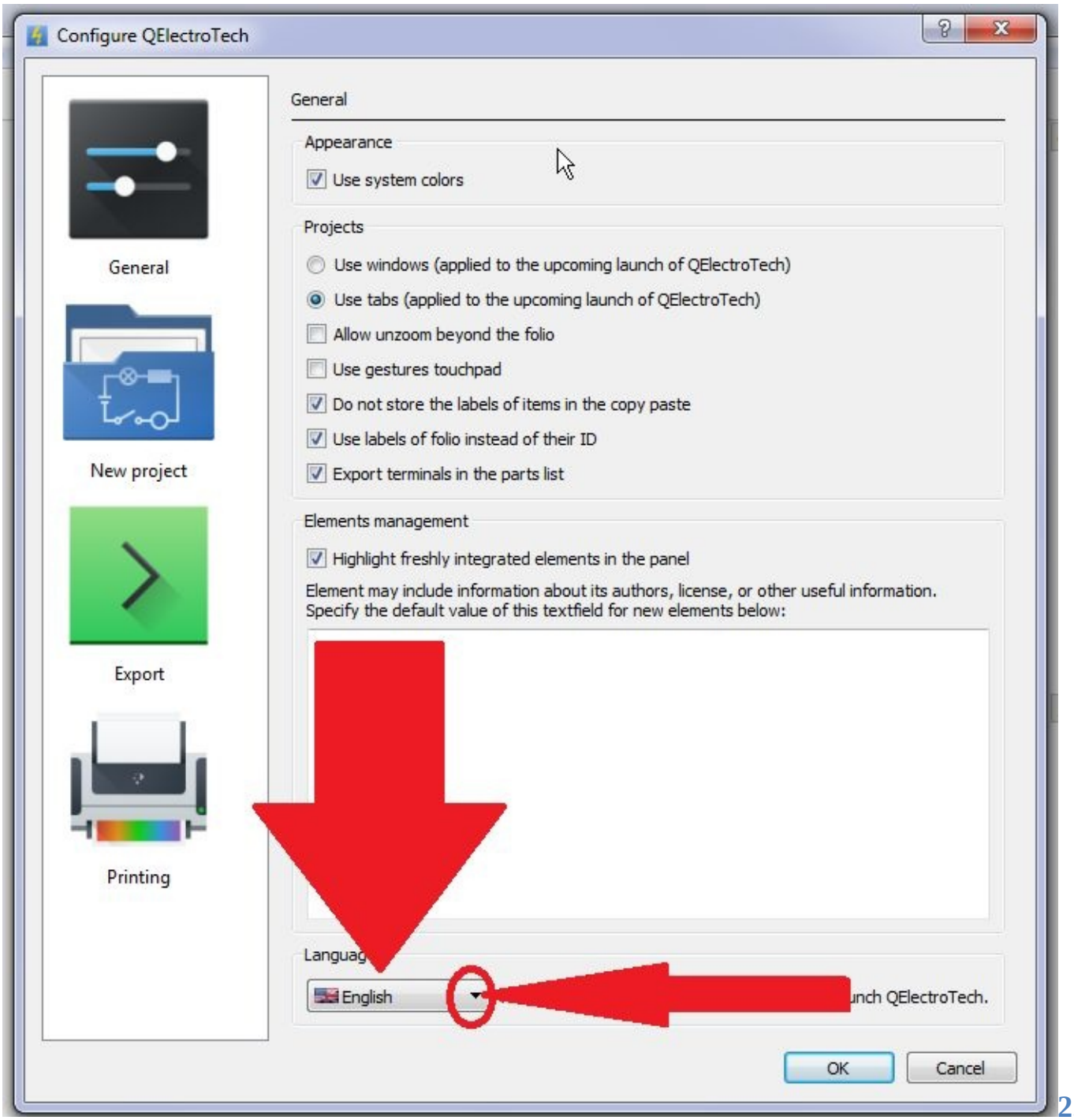
To launch QElectroTech: go to the start menu Programs QElectroTech and click on QElectroTech:



To change the language: go to Settings Configure QElectroTech



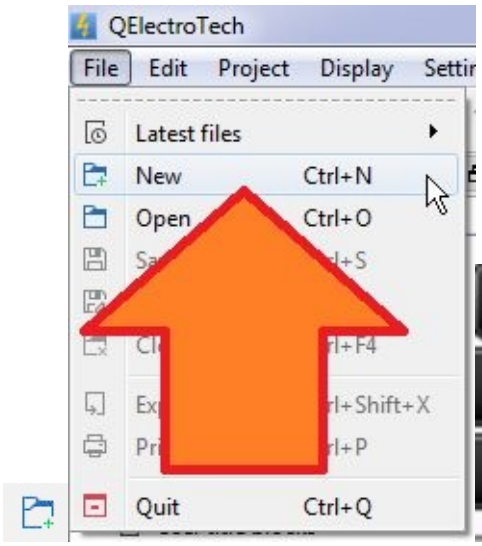
And chose your language:



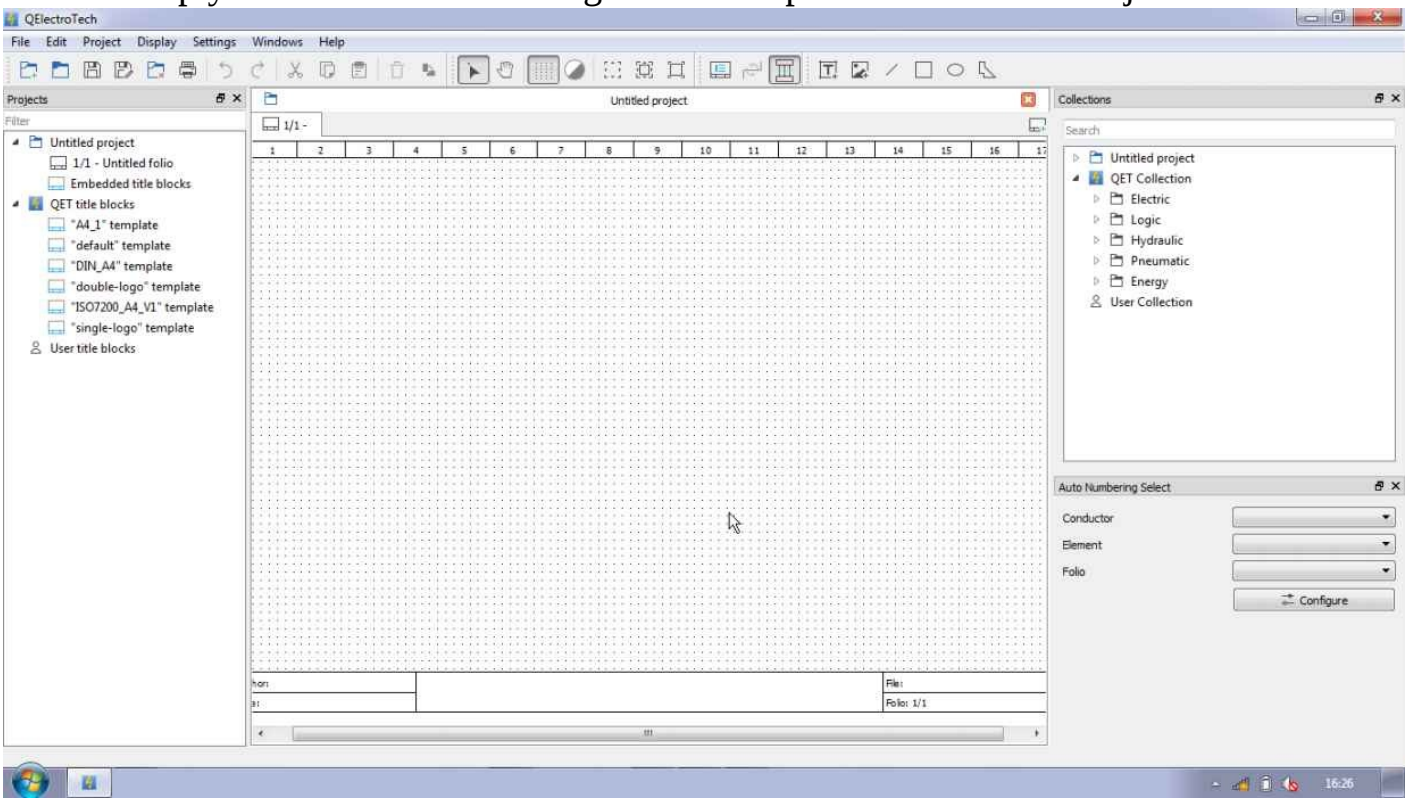
2.2

Create a new drawing project: To create and open a new drawing project:

click on the following icon
or go to the "File" menu and click on "New"
or just click on Ctrl + N:

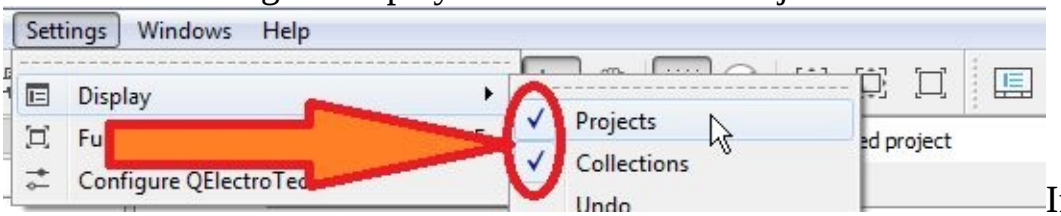


2.3 New Folio: To have an empty folio with the following default template create a new Project.

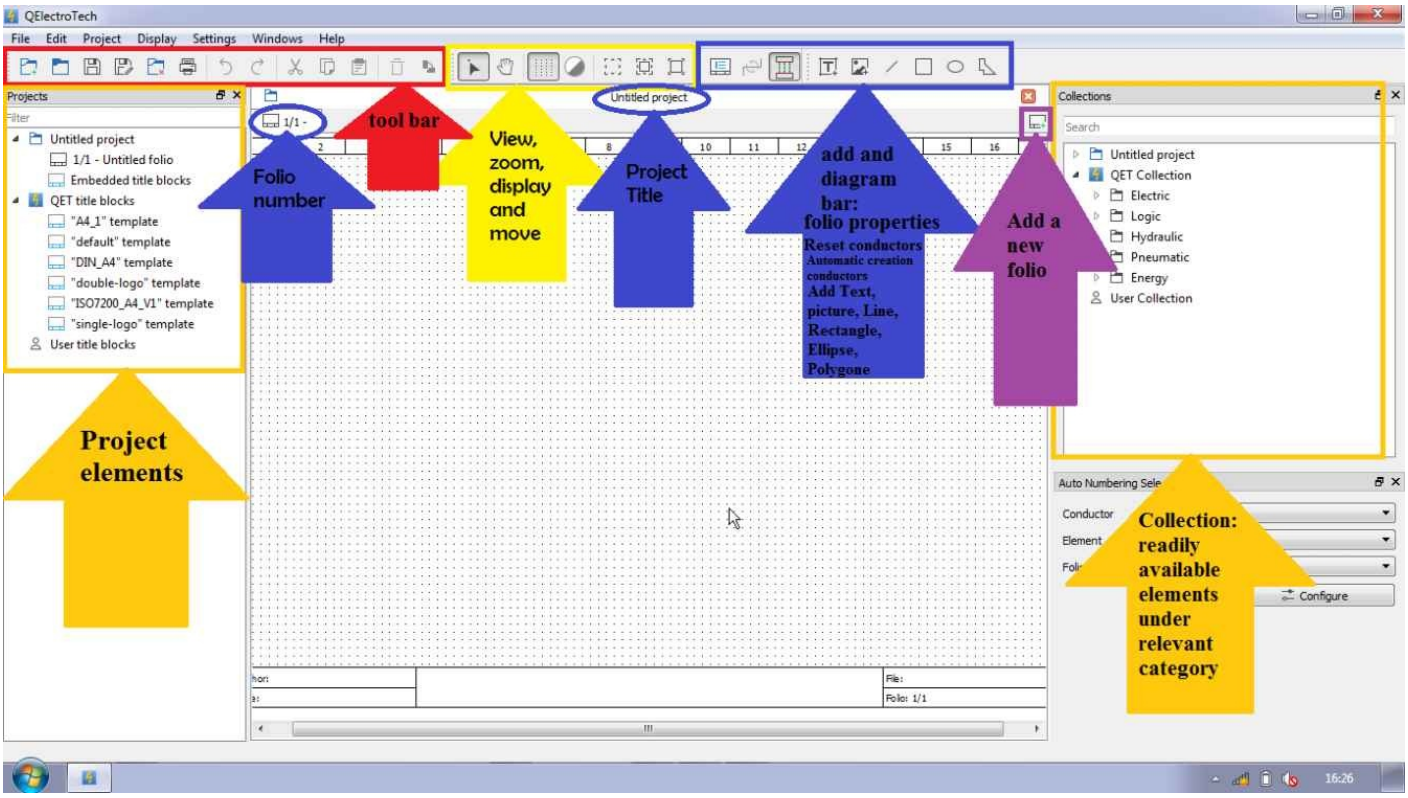


2.4 Display the Resource Elements Panel:

Go to the "Settings✉Display "and check that" Projects "and "Collections" are checked.



If this is not the case, click on "Projects" and "Collections" in the "Settings ✉ Display" to view the resources items:

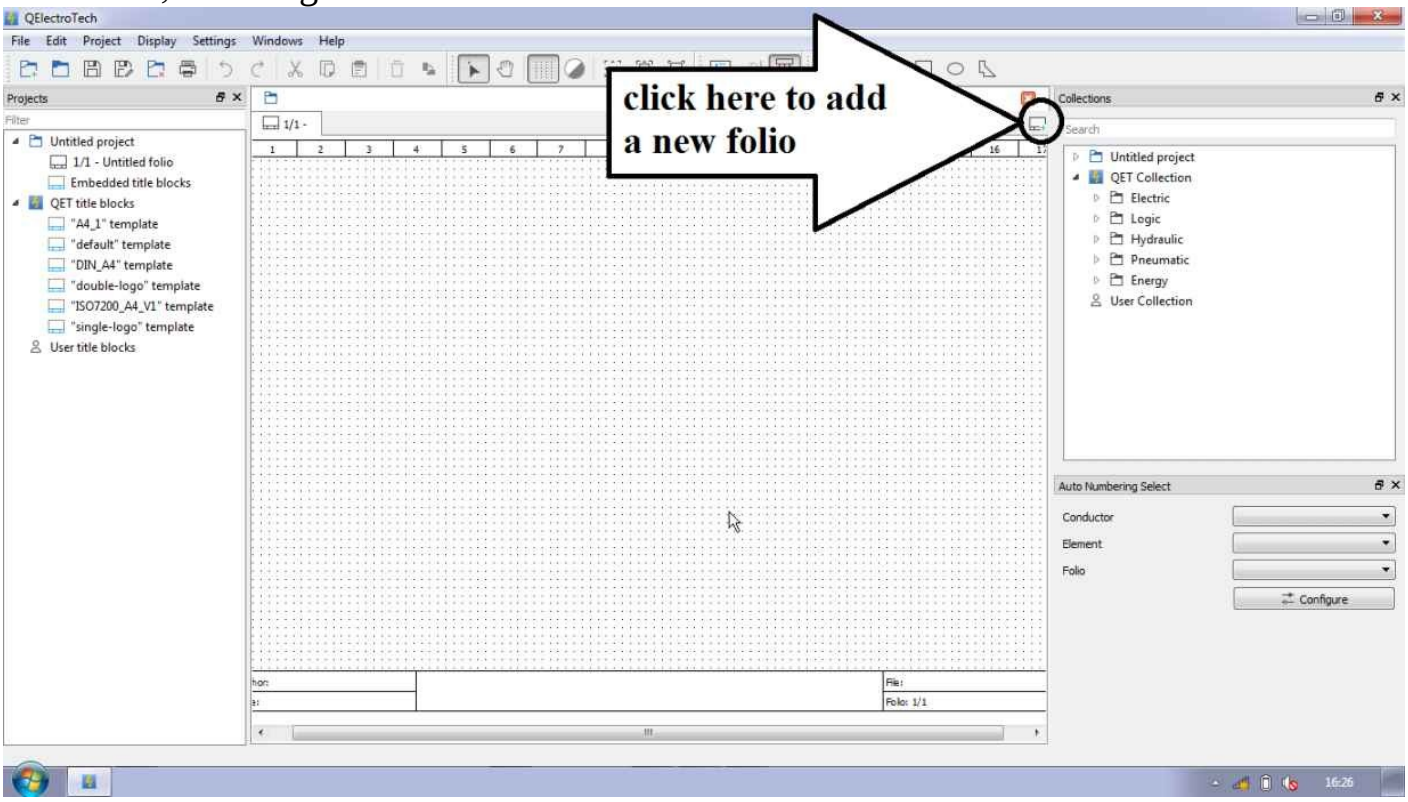


New drawing project Main Window

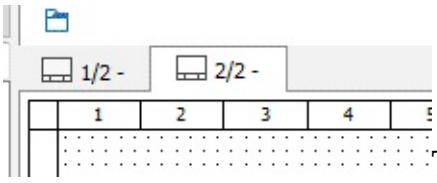
2.5 Add a new:

To add a new folio, click on the icon

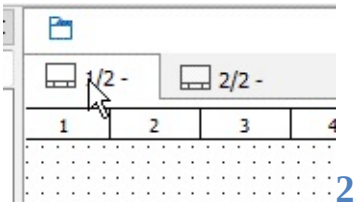
, in the right.



A new folio is added to your project.



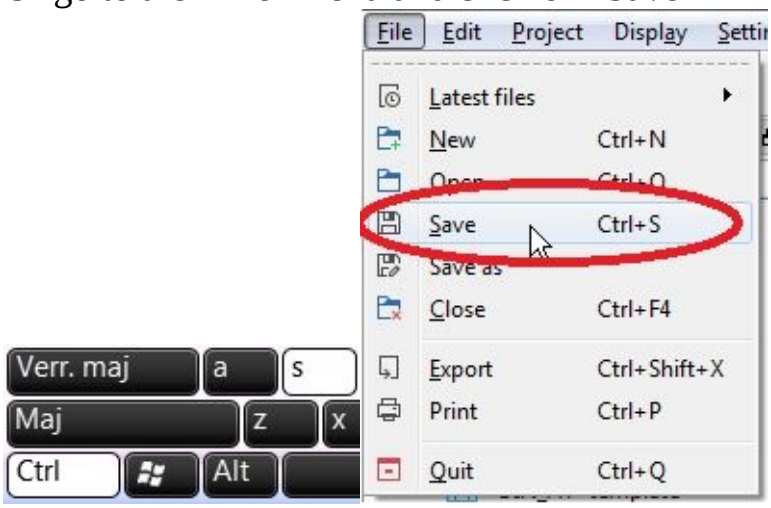
To navigate between folios click on the title of the folio



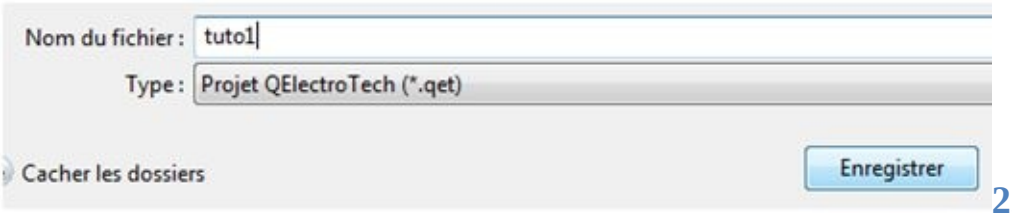
2.6 Save: To save your project:

Click on Ctrl + S

Or go to the "File" menu and click on "Save"

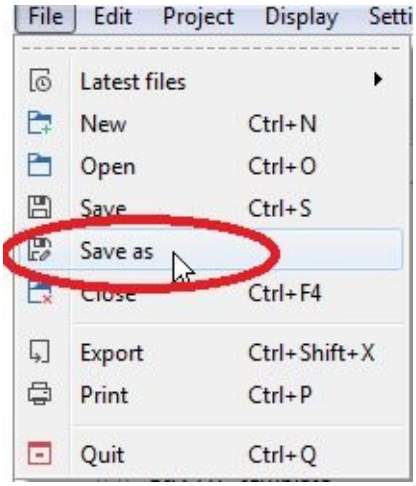


Choose the location where you want to keep the project, name it (for example: tuto1) and click Save.

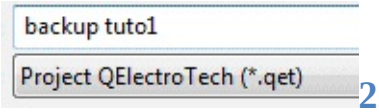


2.7 Save-as:

To save your project under another name go to the "File" menu and click "Save As"

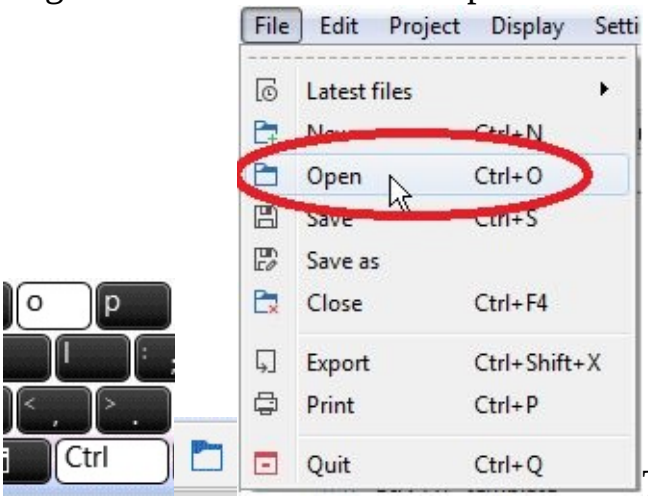


Choose the location where you want to save the project, name it (for example: backup tuto1) and click Save.



2.8 Open:

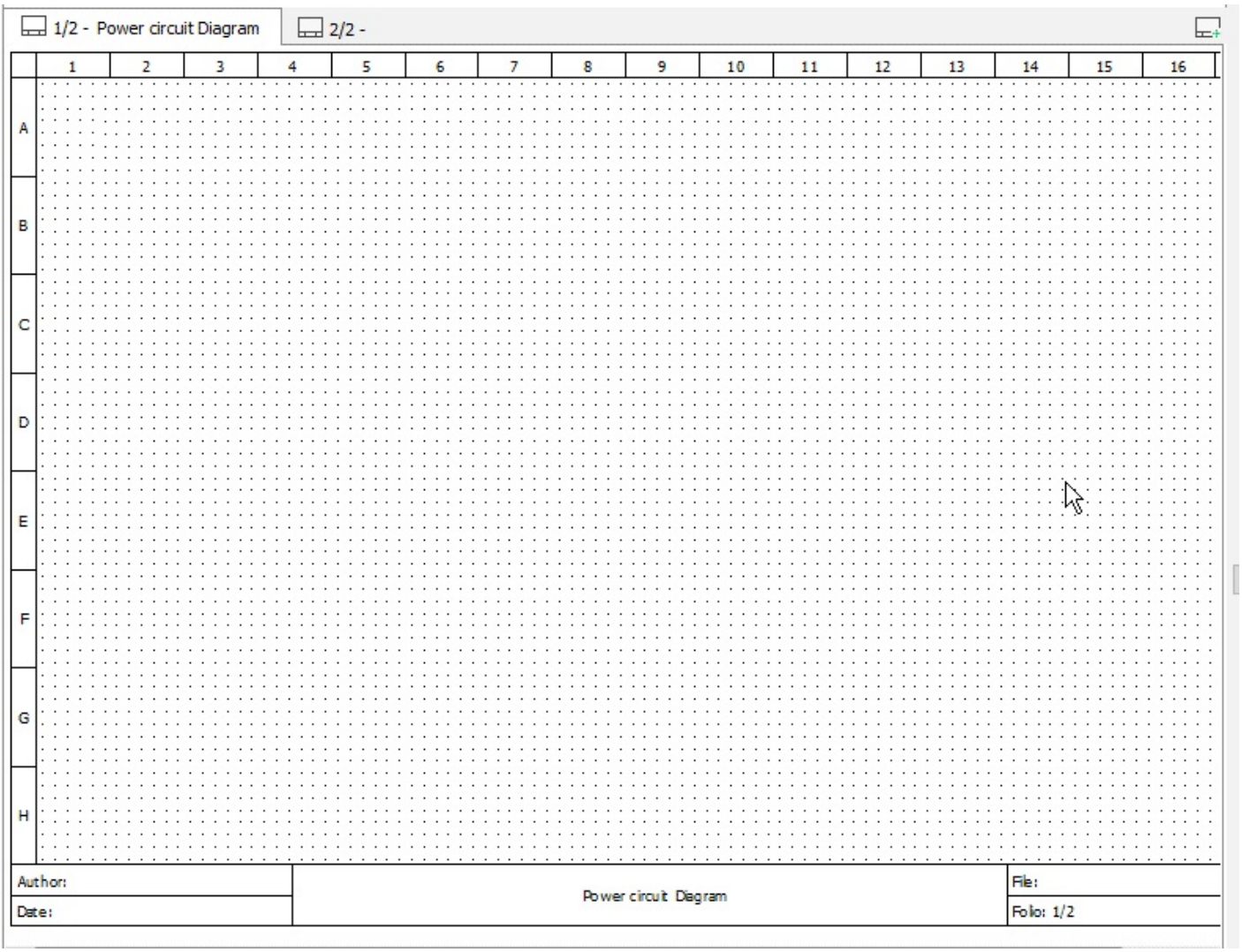
To open an existing project:
Click on Ctrl + O
or on the icon
or go to the "File" menu ✉ "open"



Then open the project location, select the project you want and click Open
Now, to continue, please open the project "tuto1" that you have saved before.

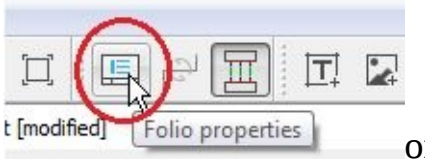
2.9 Change the title of the folio:

How to obtain the following result (a folio with the title "Power circuit Diagram"):

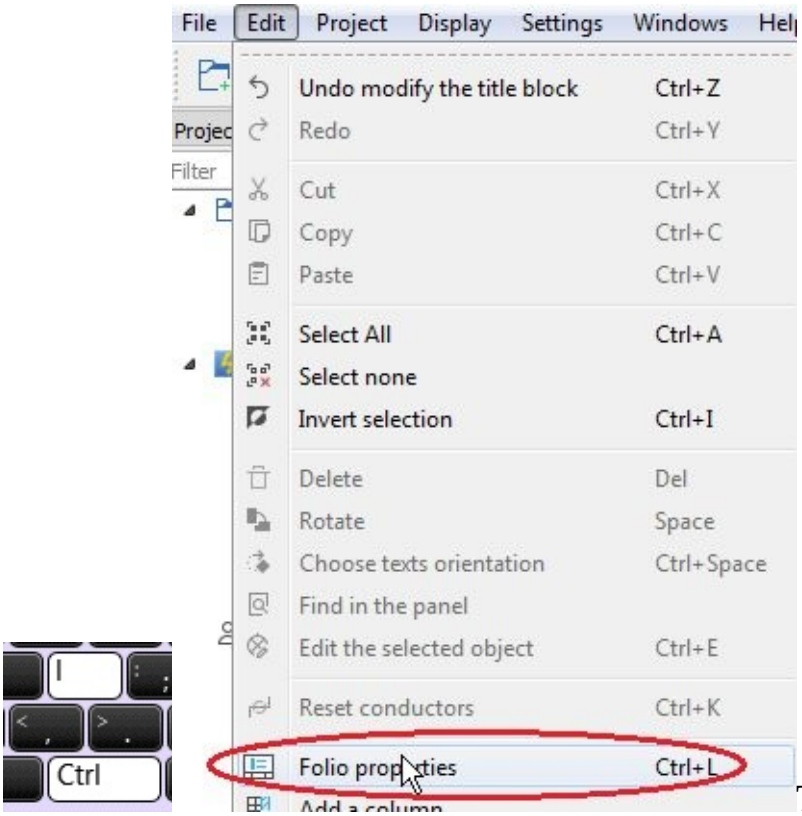


We will modify the title of the folio and name it "Power circuit Diagram", for that:
click on the following icon

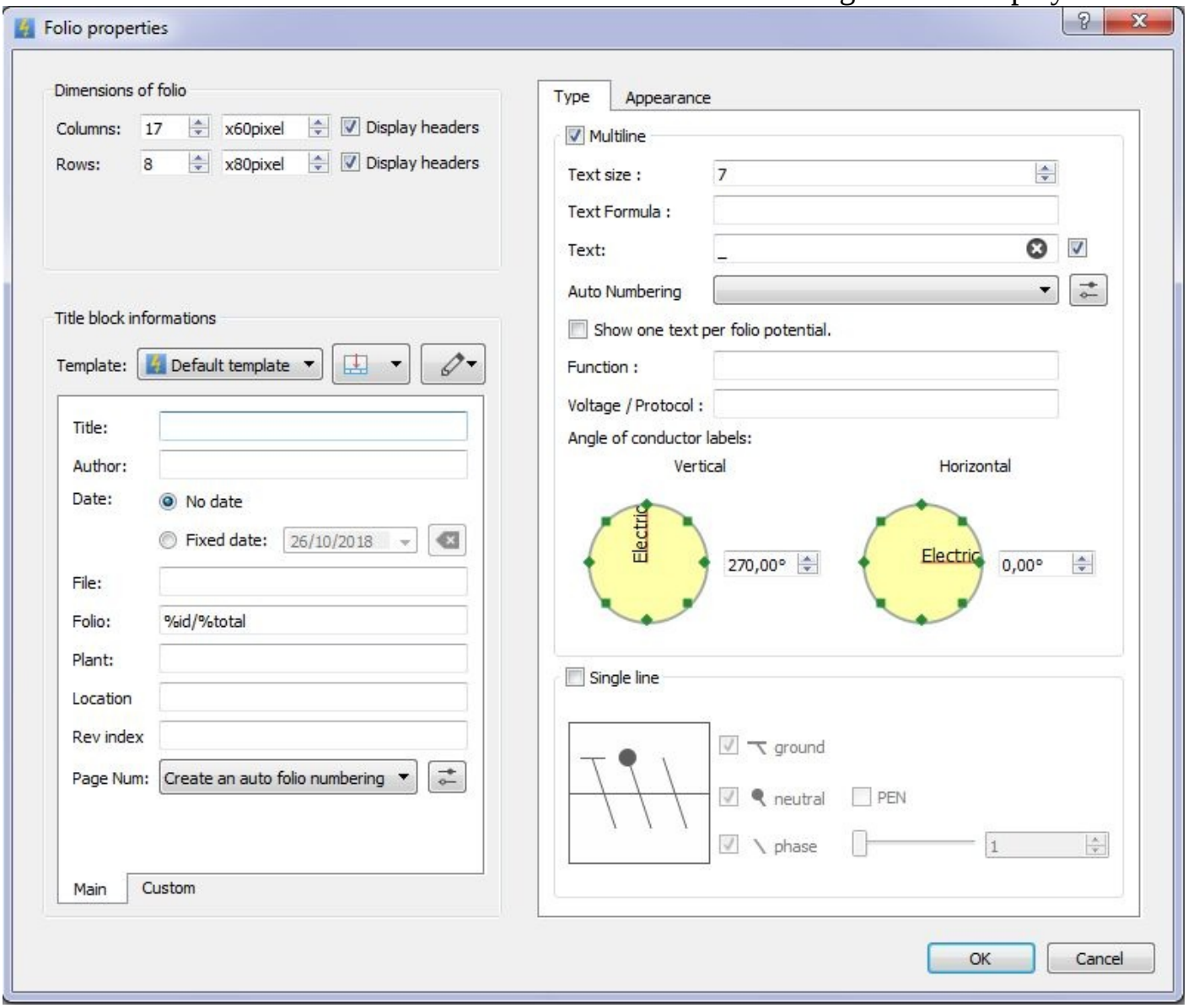

in Diagram bar



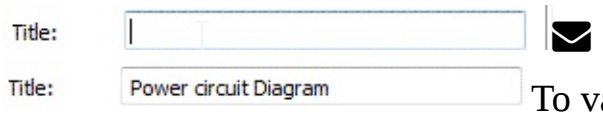
or go to the menu "Edit" -> "Edit properties of the folio"
or just click on Ctrl + L



The following menu is displayed:



Type 'Power circuit Diagram' in the 'Title' input box to change it

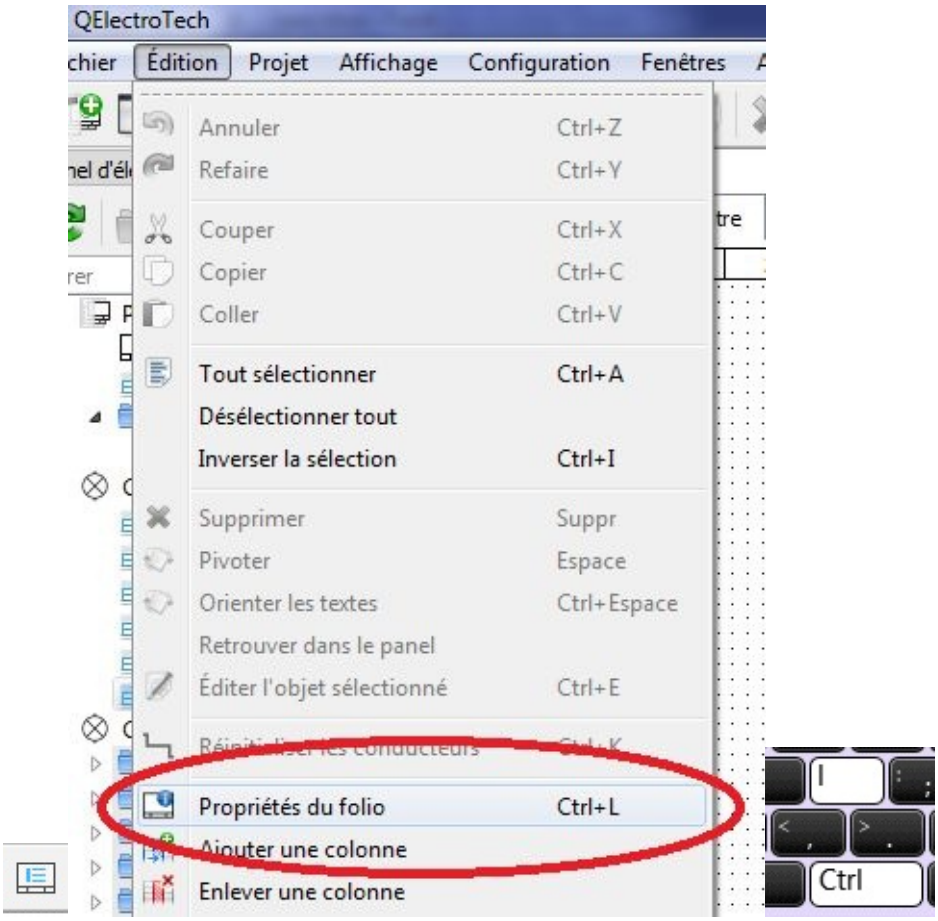


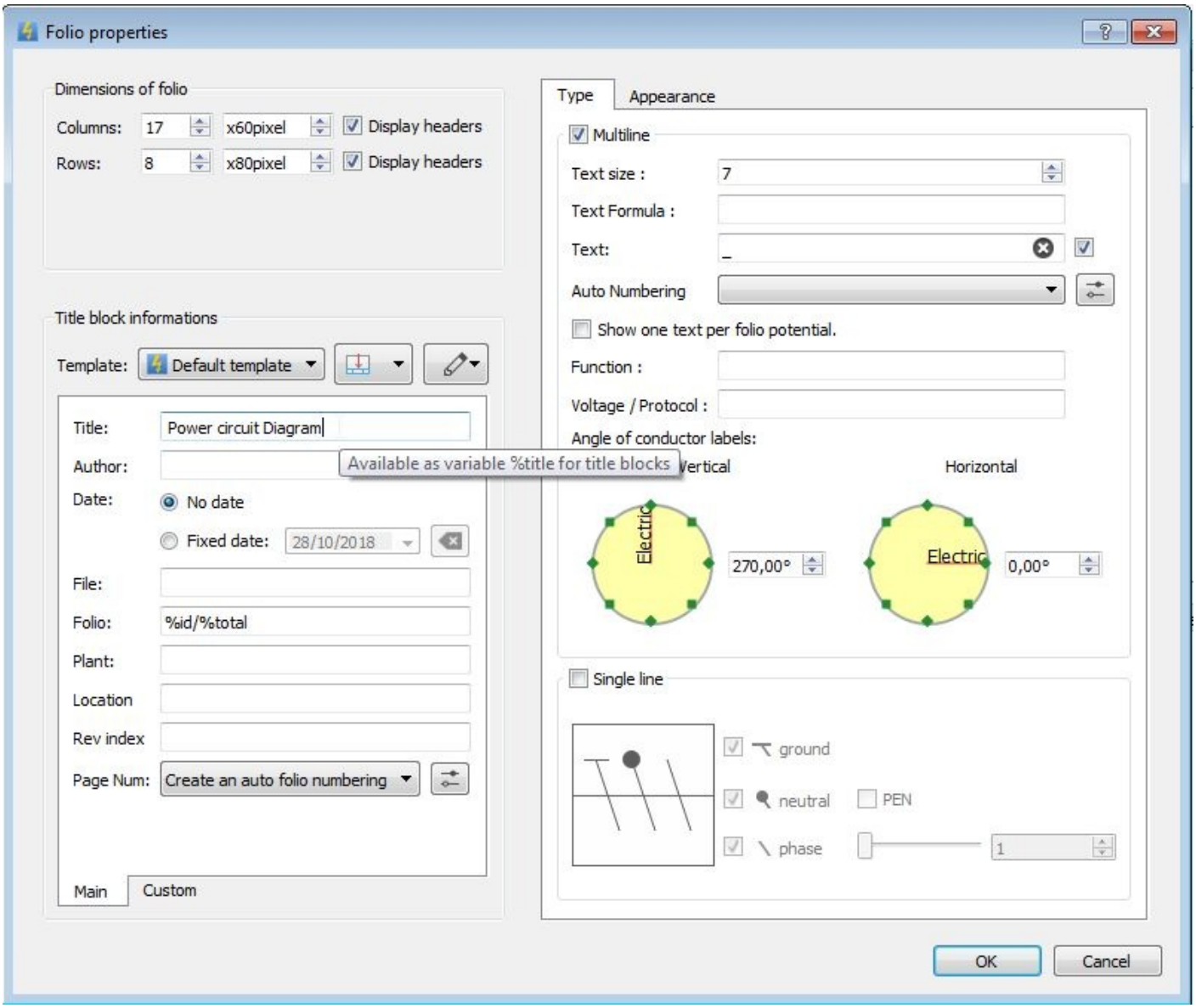
To validate the change click OK, if you do not want to change the properties of the folio click on Cancel.

2.10 Properties of the folio:

To modify the Properties of the folio:
click on the following icon
Or go to the "Edit" Editing Folio Properties menu
or just click on Ctrl + L

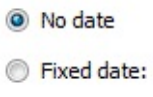
And the following menu appears:



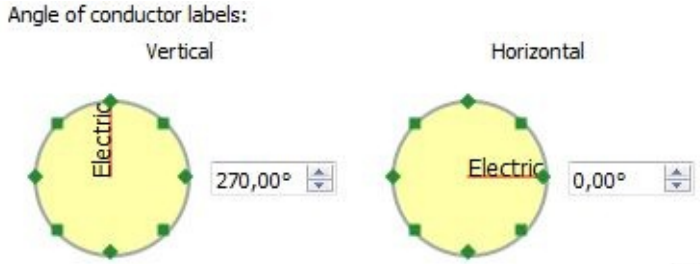


You can then change the number of columns and rows in the folio.
the default numbers are: 17 columns and 8 rows

you can also change the date and choose a fixed date



and / or change the size of the text and its orientation

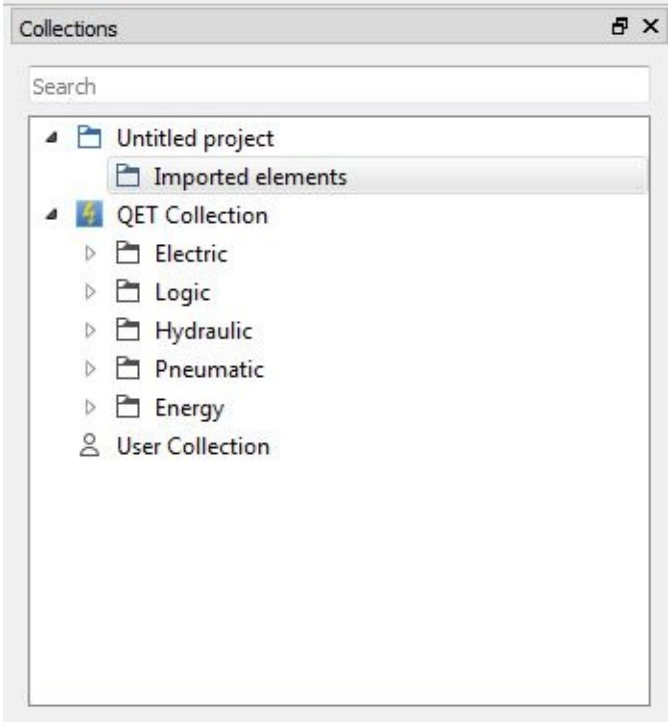


For now, type your name in the Author entry

box

2.11 The Collection & Search:

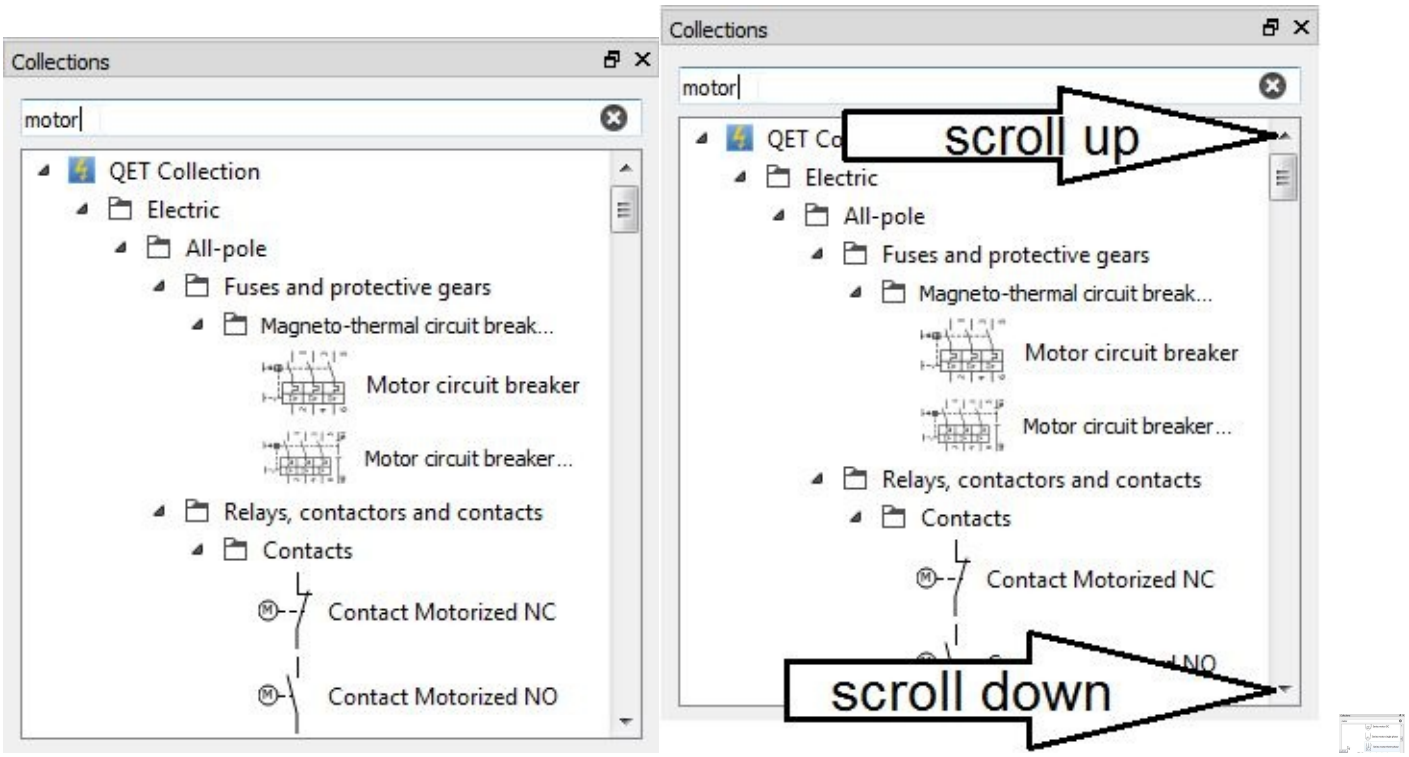
The most interesting part of the software is Collection; you will find readily available elements, imported elements and the elements created by the user.



Here's how to search for an element: type the name of the element in the search box:
What I liked the most about QElectrotech is the ability to edit the elements in the way that suited

me or according to the needs of customers.

Example:
Search for Motor: type motor and press enter, and scroll to find the Motor element



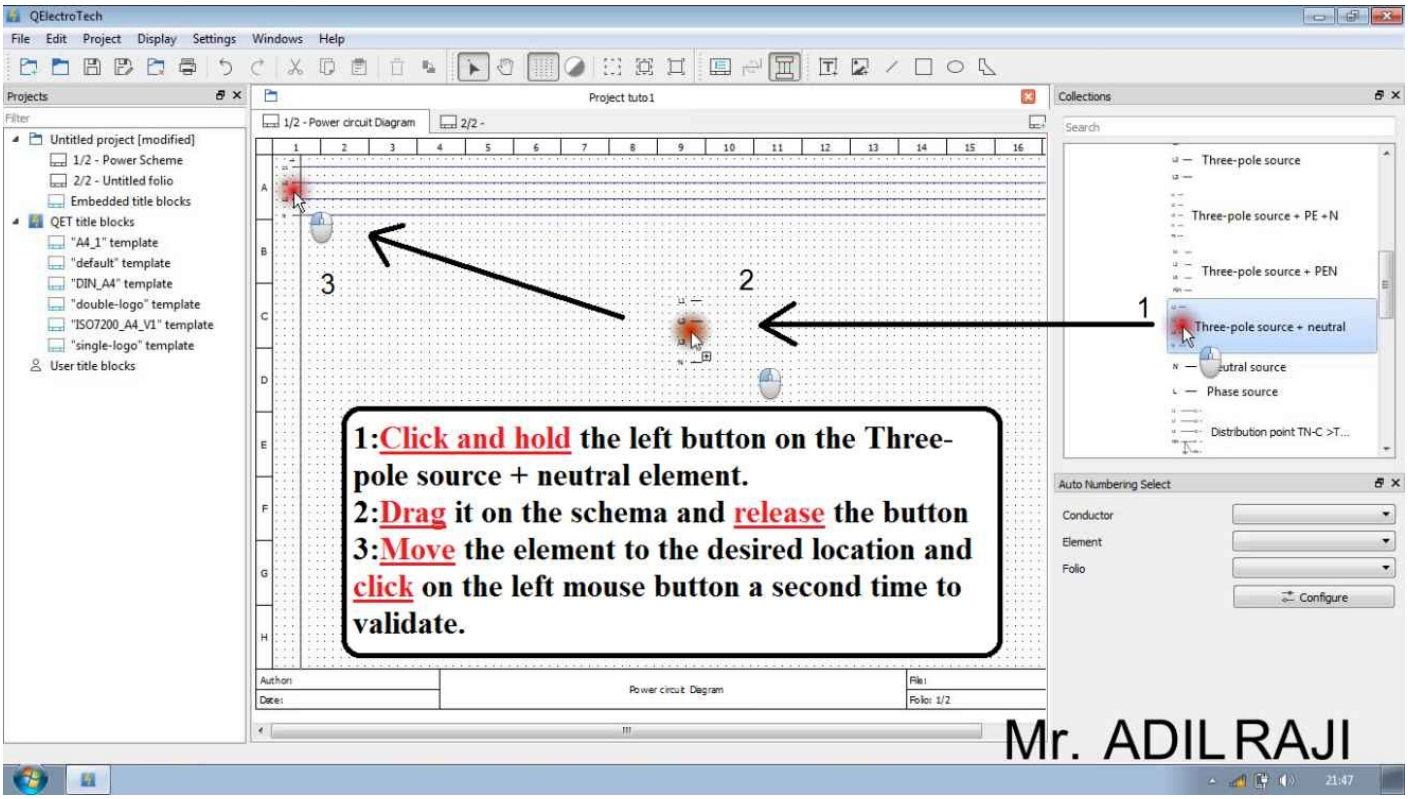
2.12 Insert and link automatically elements:

The insertion is done by a simple "drag & drop and place" on the drawing work space.

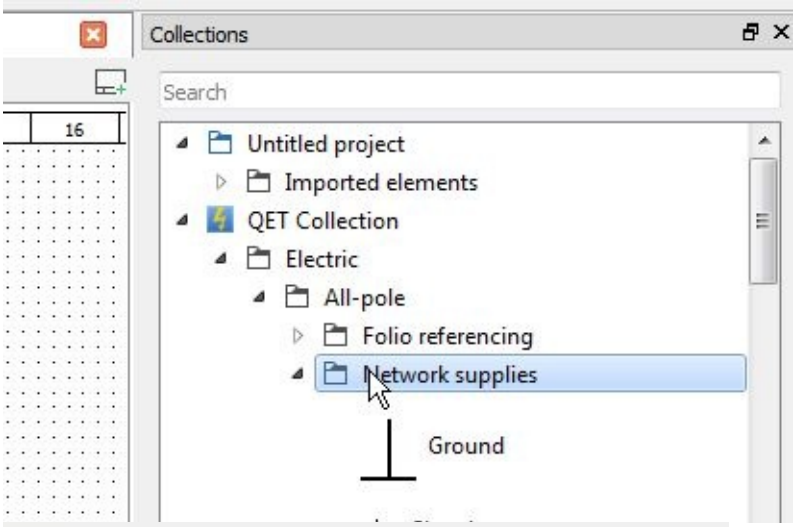
So if you want to insert an element in the drawing work space, look for this element in the Collections, click and hold the left button on the chosen element, drag it on the drawing work space, release the left button, move the element to the desired location and click on the left mouse button a second time to validate it, repeat the last step as many times as necessary; to finish click on the Escape button (ESC) on your keyboard.

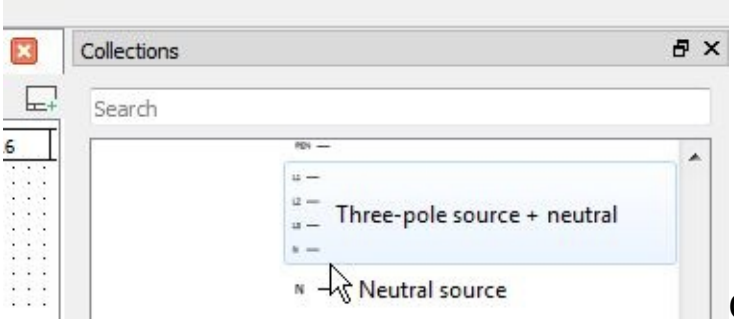
2.12.1 Example:

We will insert in the following example a three-pole source + neutral (L1, L2, L3 and N). Open the project "tuto1" as described in "**2.8.Open**".

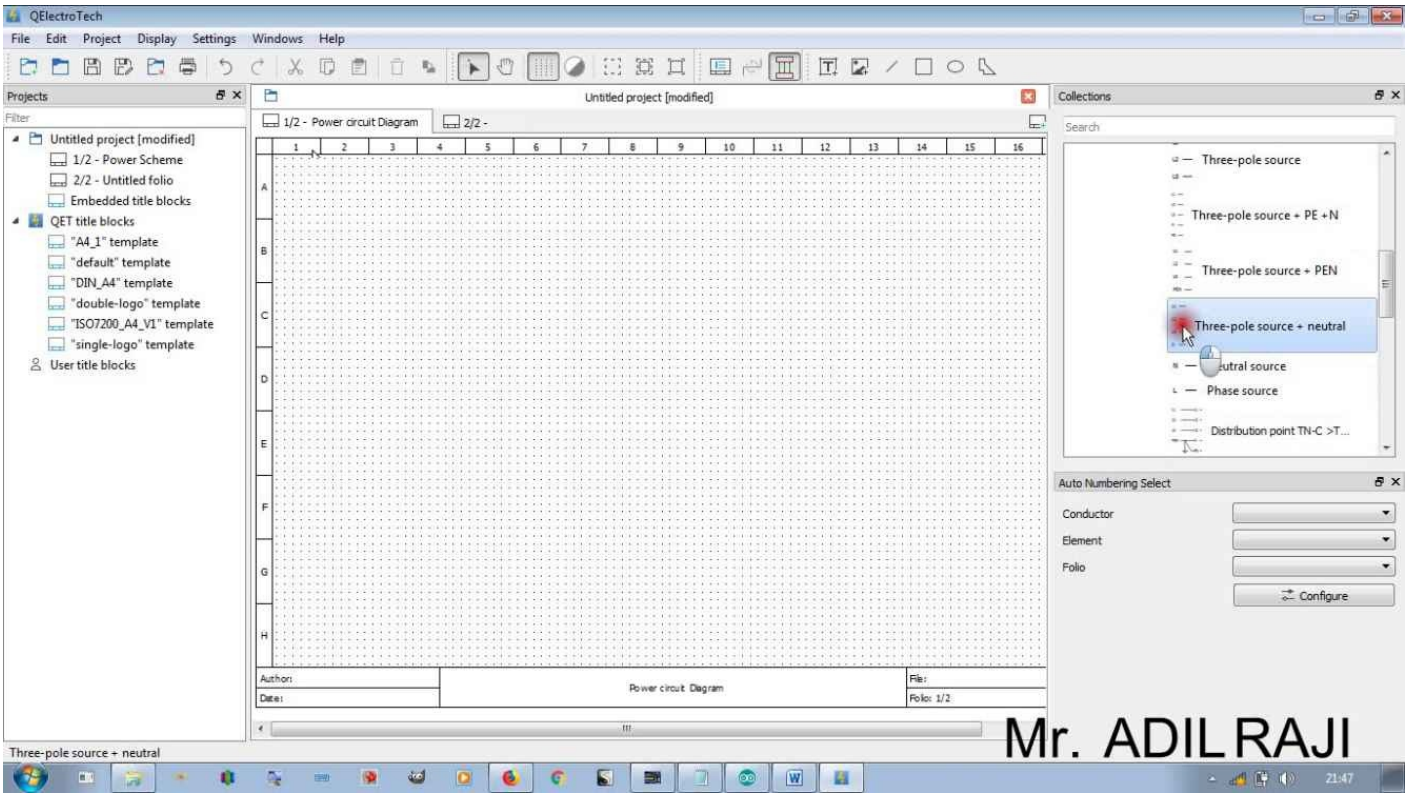


For this go to "Collections" ✉ "Electric" ✉ "Allpole" ✉ "Network supplies
Scroll down and look for "Three-pole source + neutral"

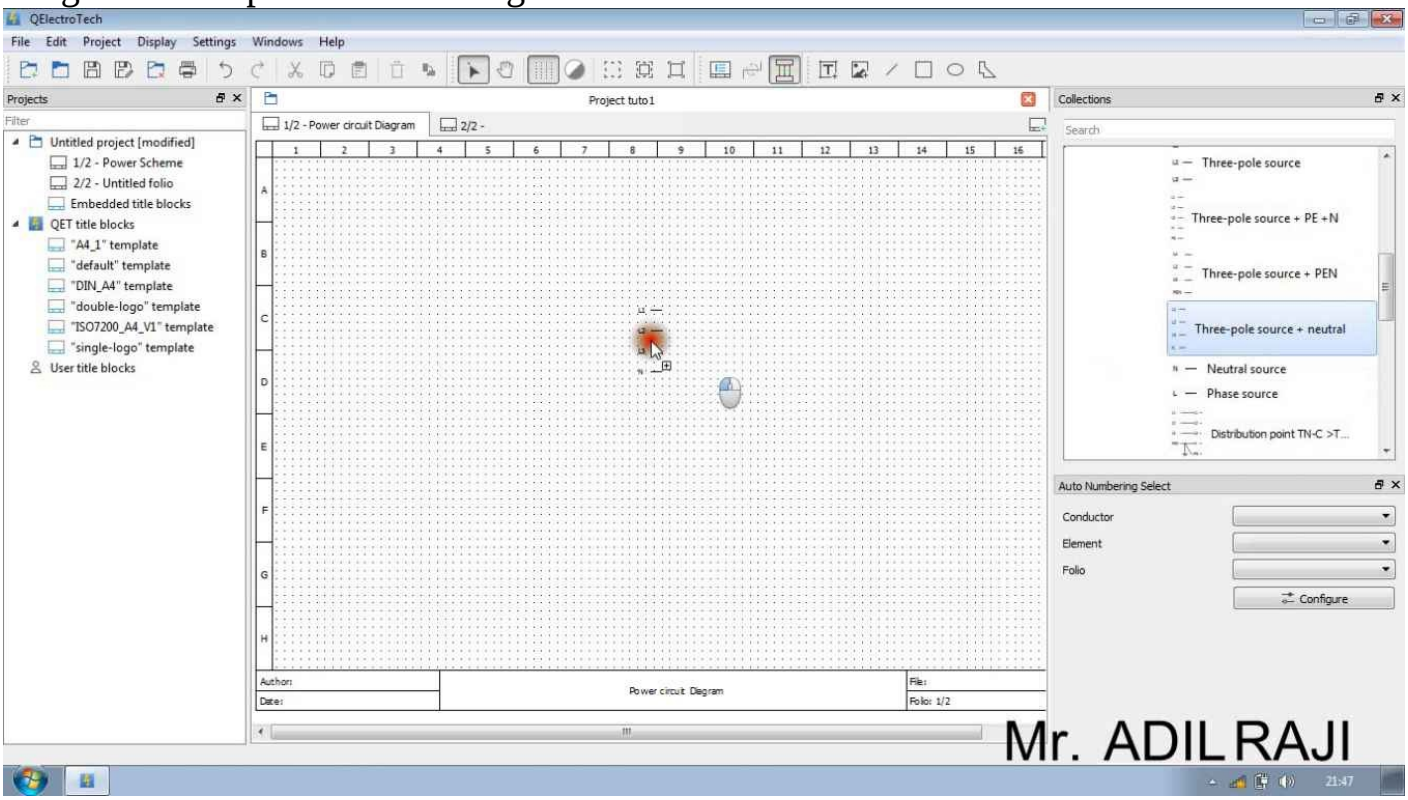




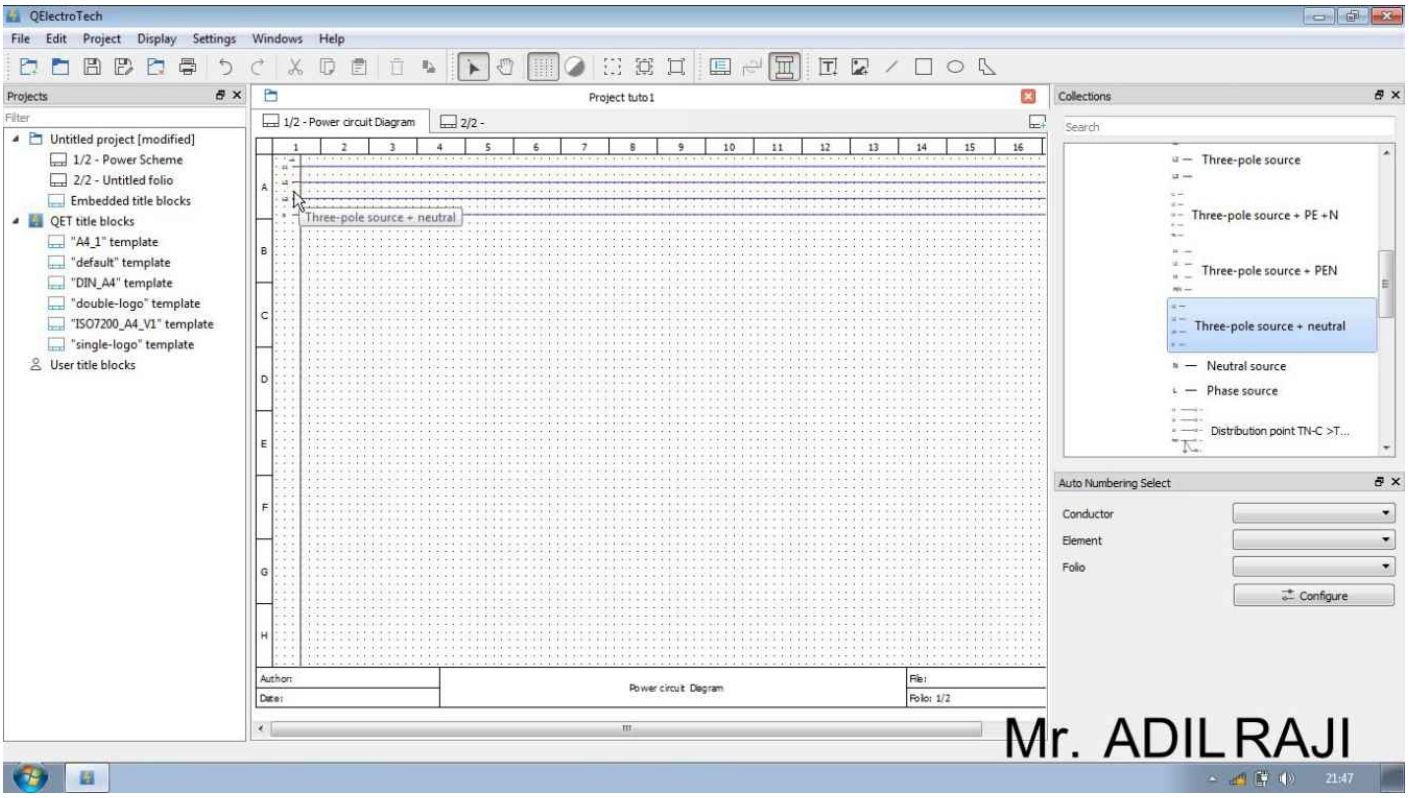
Click and hold the left mouse button;



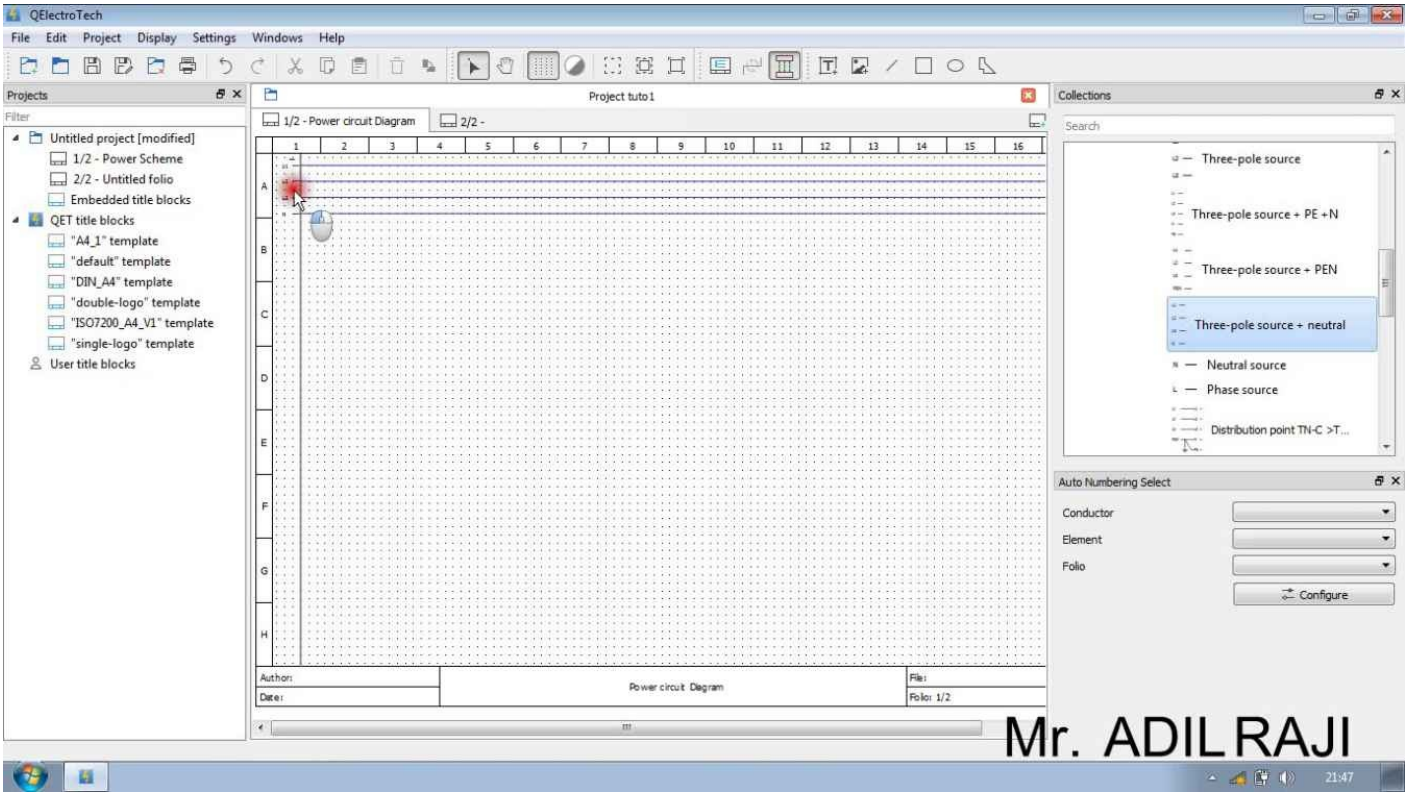
Drag it into the power circuit diagrams:



Release the left button and move the item to the desired location



And click the left mouse button to validate the location.



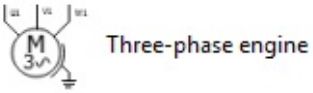
For more explanation we will in the rest of this chapter realize a direct online (DOL) starter circuit diagram of induction motor.

So for the power diagram we will insert and link the following elements:

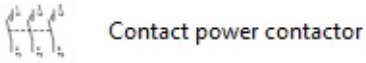
Real name of the component

Name of the component to enter in the search box Filter

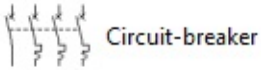
Collection containing the component

Induction motor 

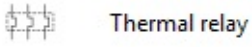
Three-phase engine

Power contactor 

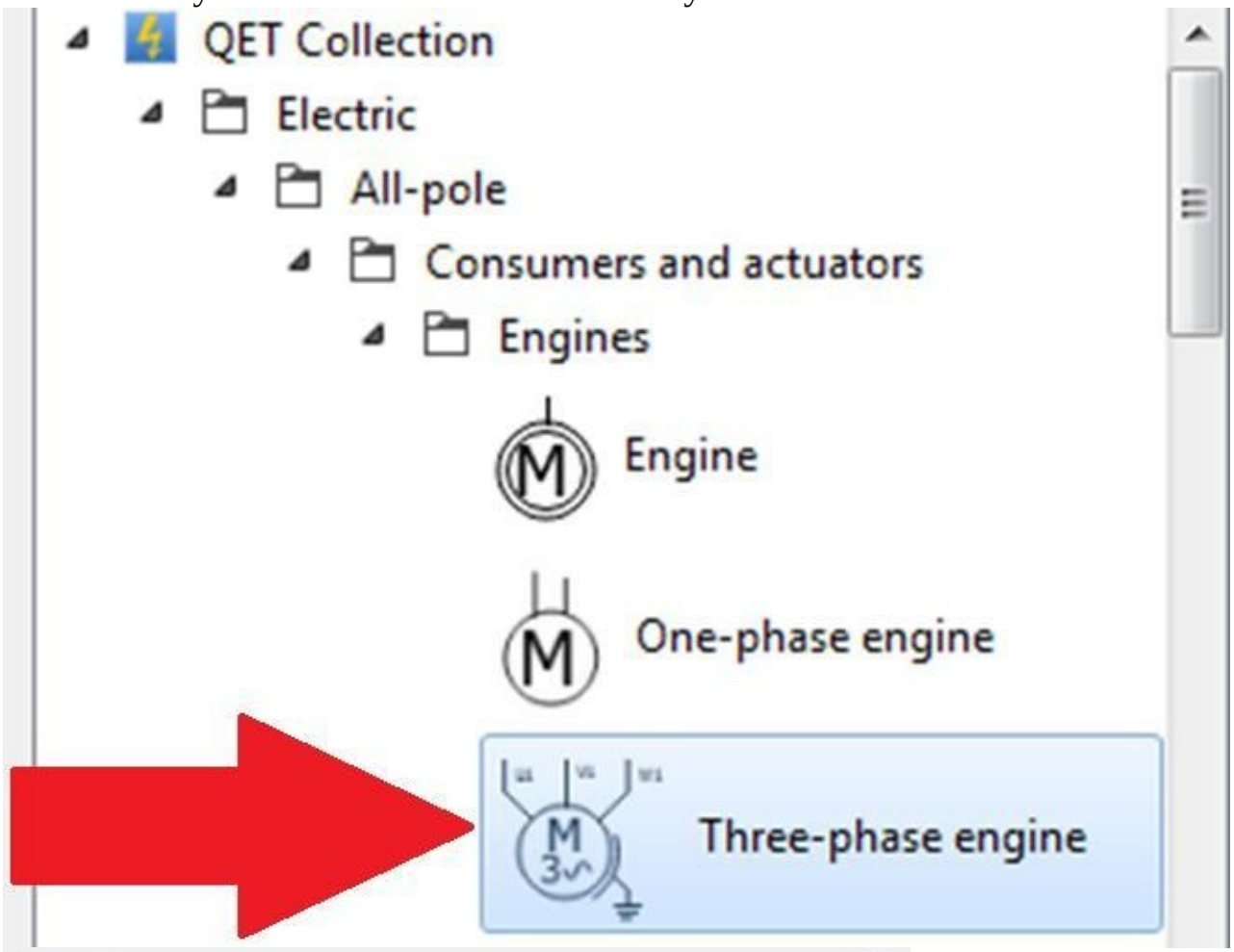
Contact power contactor

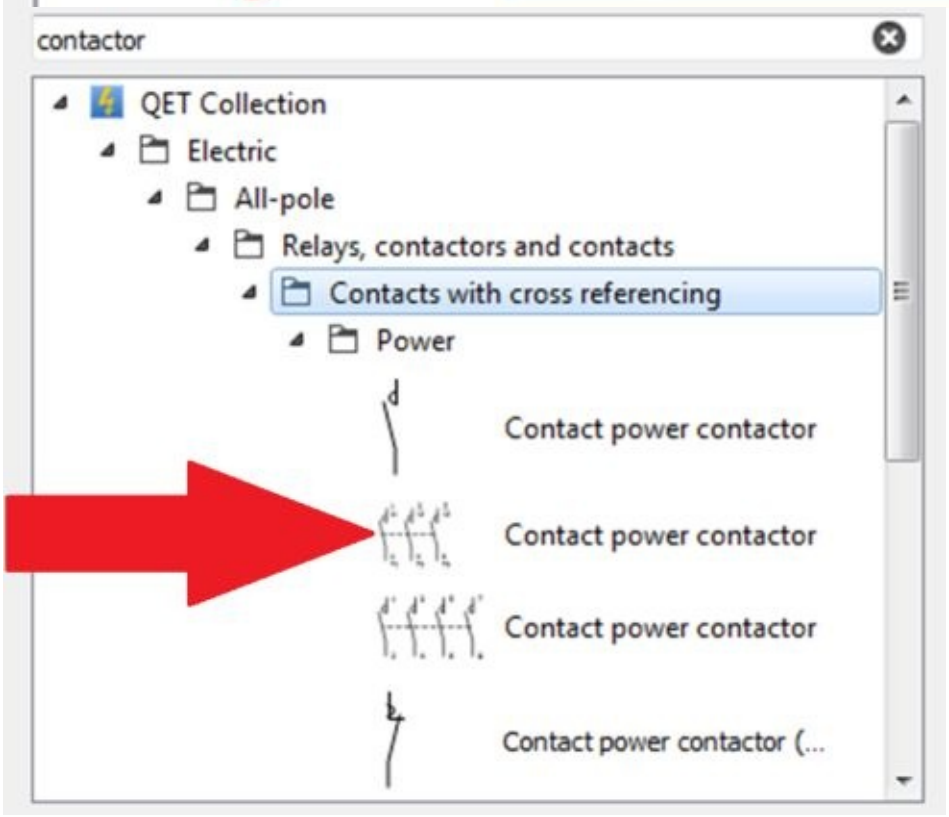
Circuit breaker 

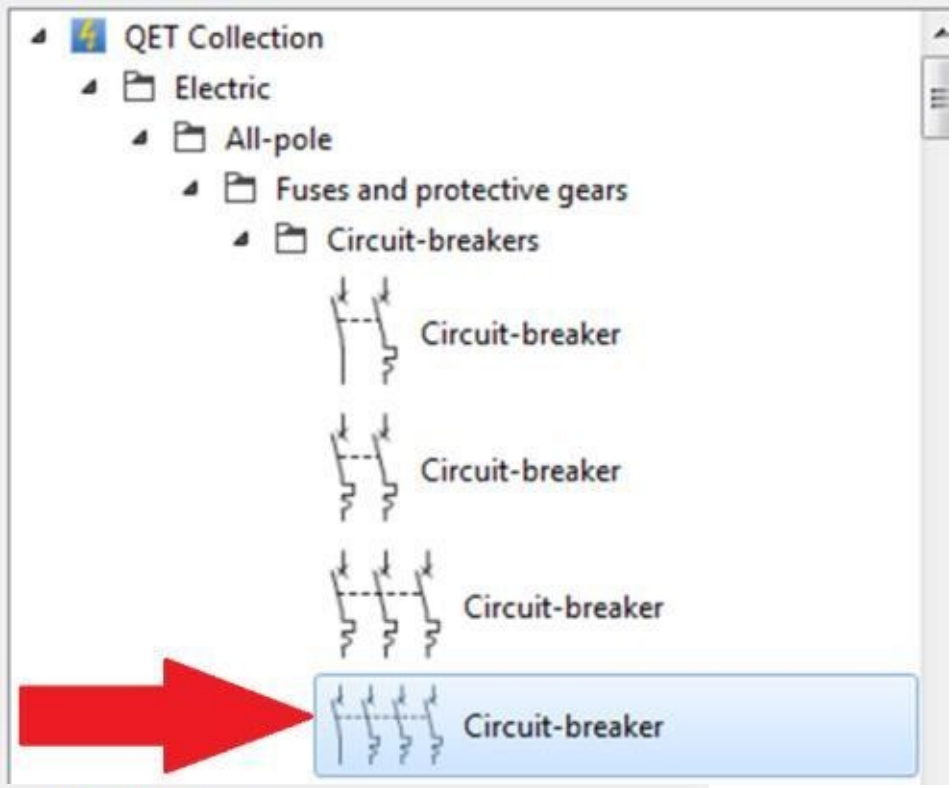
Circuit-breaker

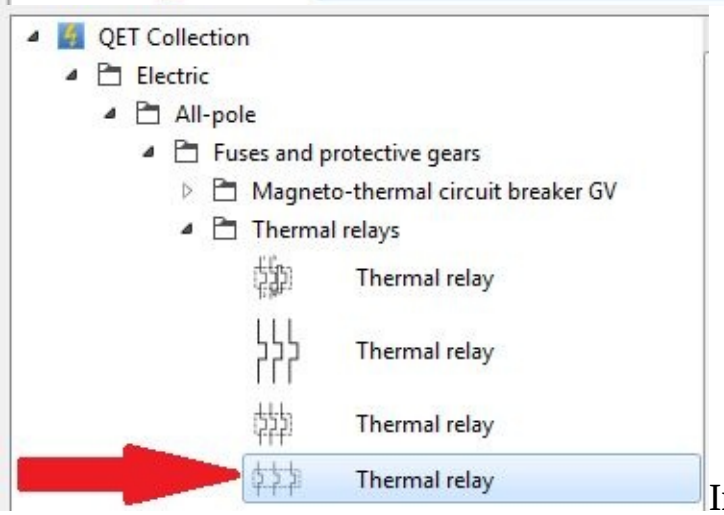
Thermal relay 

Thermal relay





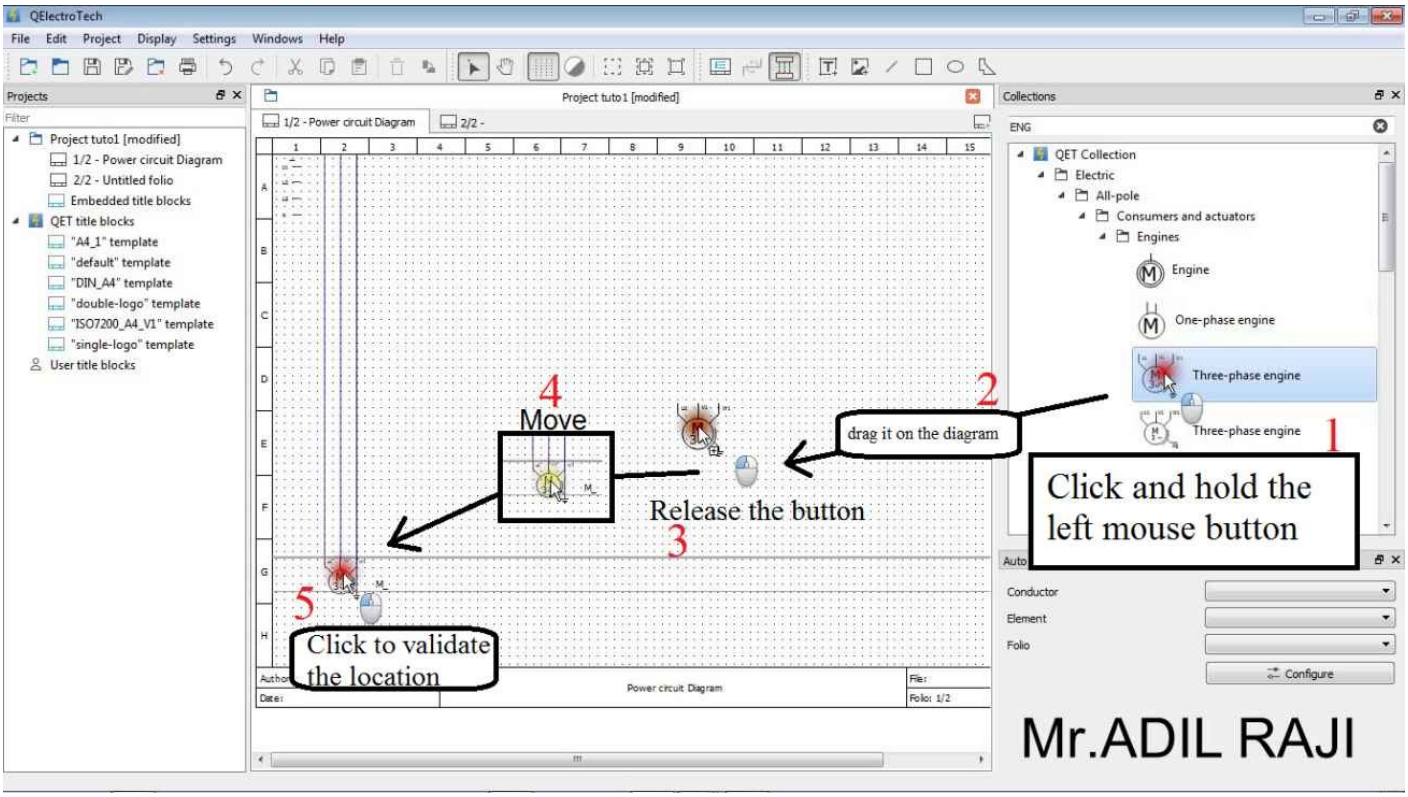




In order to realize a direct online (DOL) starter circuit diagram follow exactly the steps described below:

2.12.2 Inserting the motor:

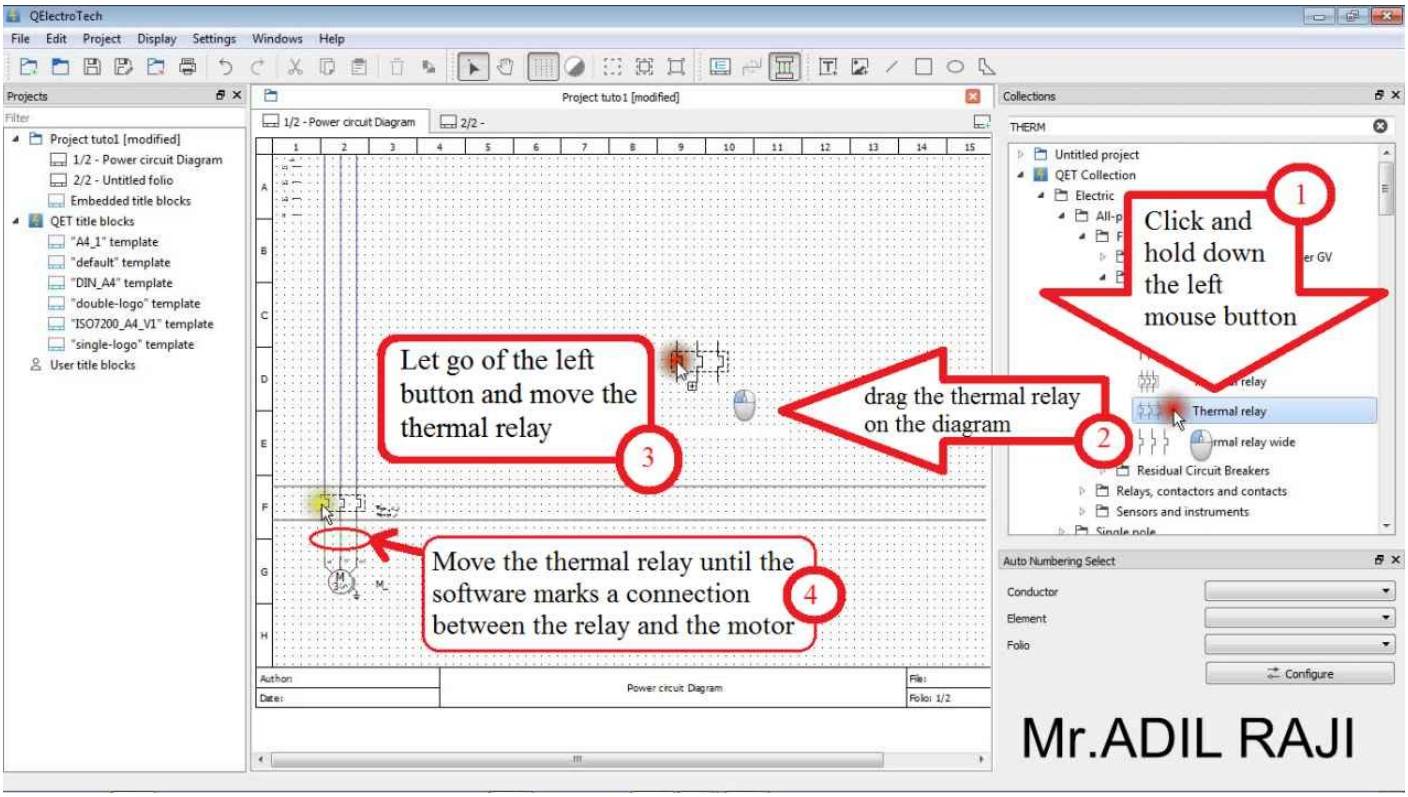
1: Click and hold the left mouse button
2: Drag it on the diagram
3: Let go of the left button (Release),
4: Move it to the desired location
5: Click on the left mouse button to validate the location.



2.12.3 Insertion of the thermal relay and automatic connections with the motor:

It is possible to connect the elements between them automatically (this connection is made on two axes: horizontal and vertical).
To show you how to do this we will insert a thermal relay and position it in such a way that the software automatically connects it with the motor.

1: Click on the Thermal relay and hold down the left mouse button
2: Drag the Thermal relay on the diagram
3: Let go of the left button.
4: Move the thermal relay until the software marks a connection between the relay and the motor.



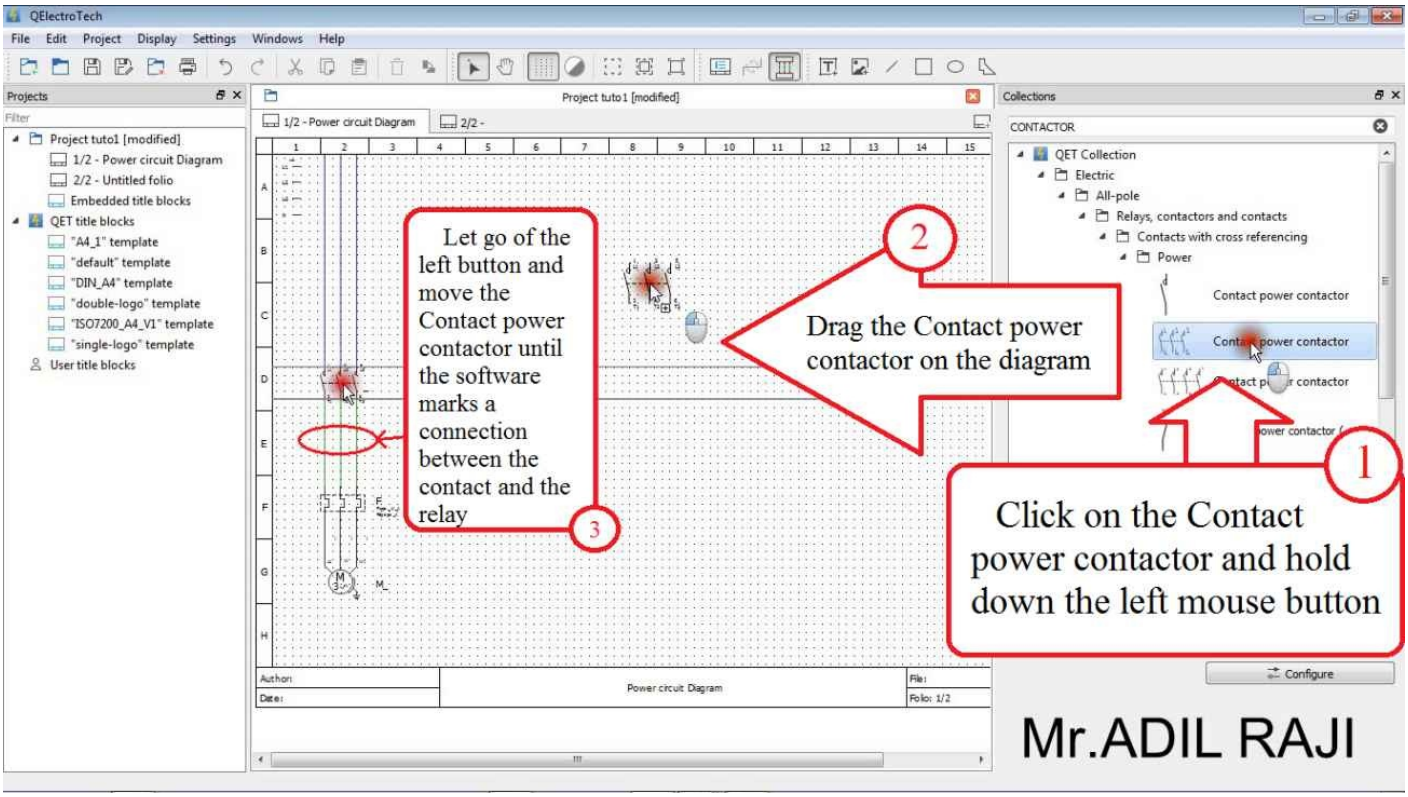
5: Click on the left mouse button to validate the location:
Finish inserting the thermal relay click on the Escape button (ESC) on your keyboard.

The escapes button (ESC)

2.12.4 Insert contact power contactor:

We will now insert power contacts and link it automatically by the same method.

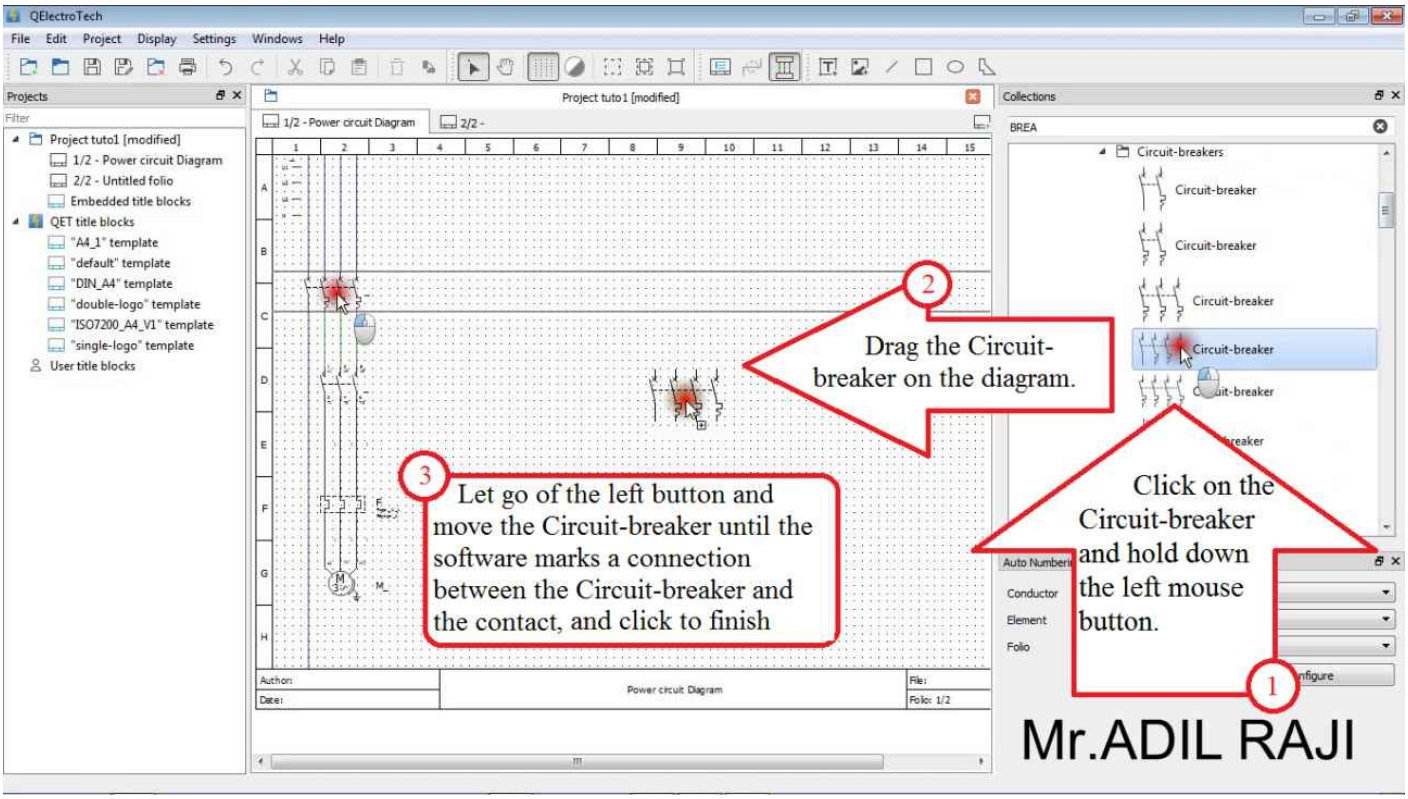
1: Click on the Contact power contactor and hold down the left mouse button.
2: Drag the Contact power contactor on the diagram.
3: Let go of the left button and move the Contact power contactor until the software marks a connection between the contact and the relay.



4: the left mouse button to validate the location:
Click on Escape button (ESC) on the keyboard to finish.

2.12.5 Inserting a Circuit-breaker

We will now insert a Circuit-breaker and automatically link it by the same method.

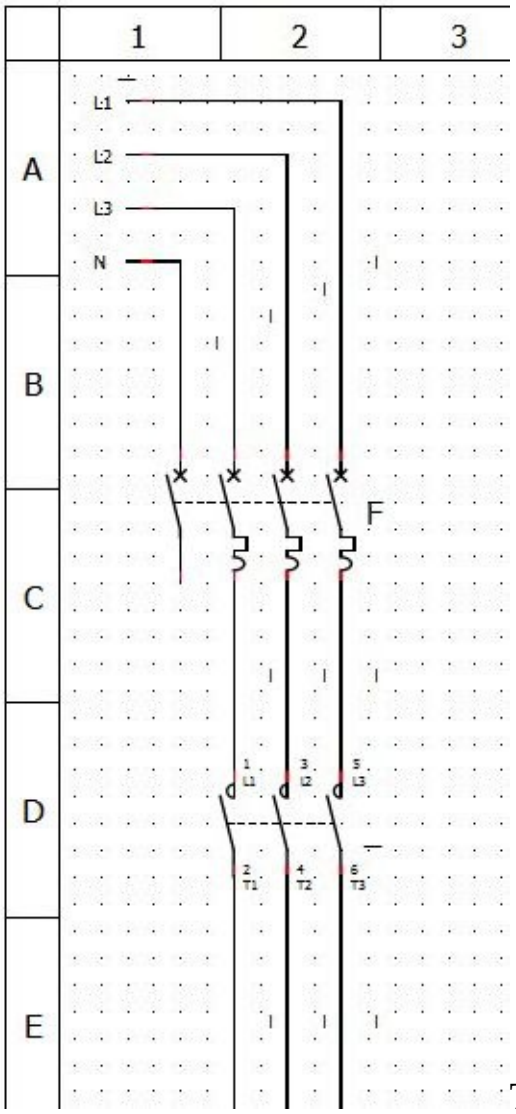


To end this exercise click on "Save"

 to save the modifications made to the project.

2.13 Manually linking the elements:

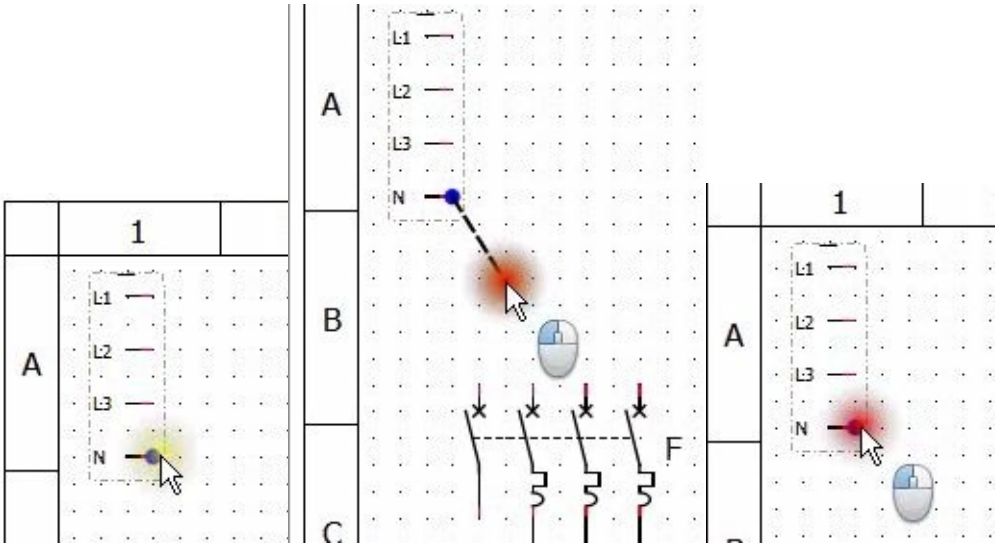
To learn how to manually link two items in a circuit diagram perform the steps in the following example to get this result:

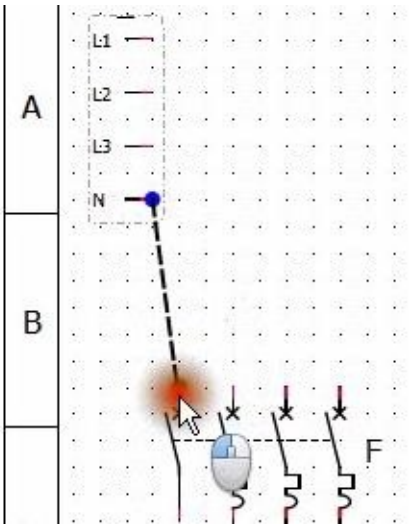


To achieve that result we will start by manually linking N with the Circuit-breaker:

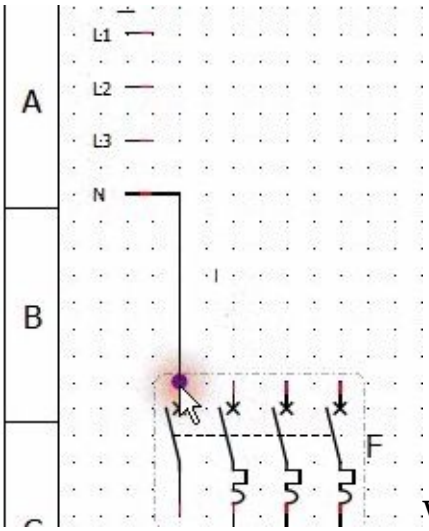
1. Move the mouse cursor until you select the end of the N source terminal, **click and hold** the left mouse button to maintain the selection

2. Move the cursor to the end of the 1st contact of the Circuit-breaker





3. Release the left mouse button to validate

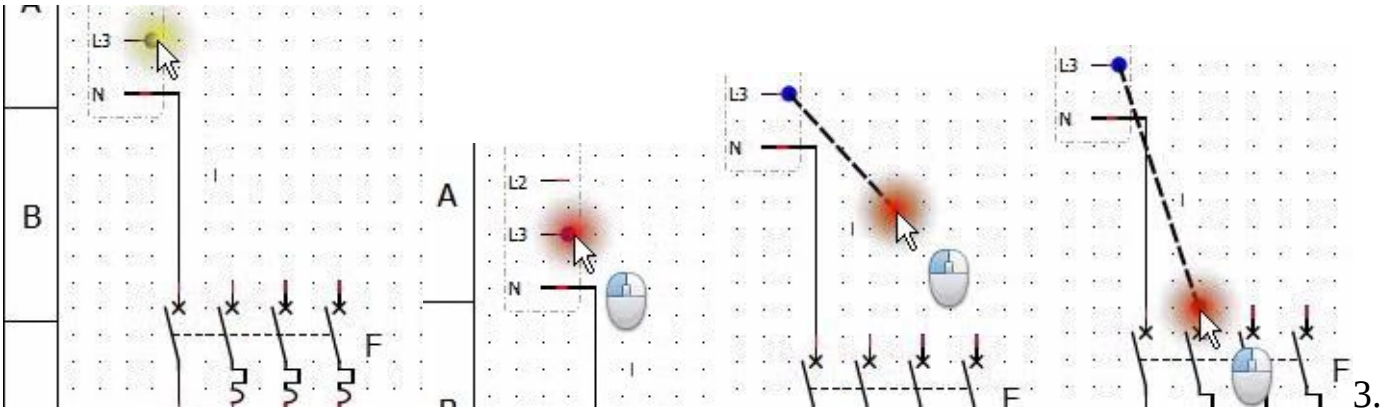


We will repeat the same steps for L1, L2 and L3.

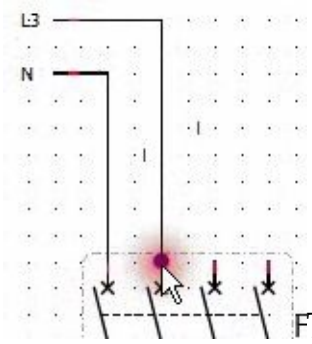
So to manually link L3 with the circuit-breaker please follow these steps:

1. Move the cursor until you select the end of the L3 terminal, **Click and hold** the left mouse button to maintain the selection

2. Move the cursor towards the end of the 2nd contact of the circuit-breaker



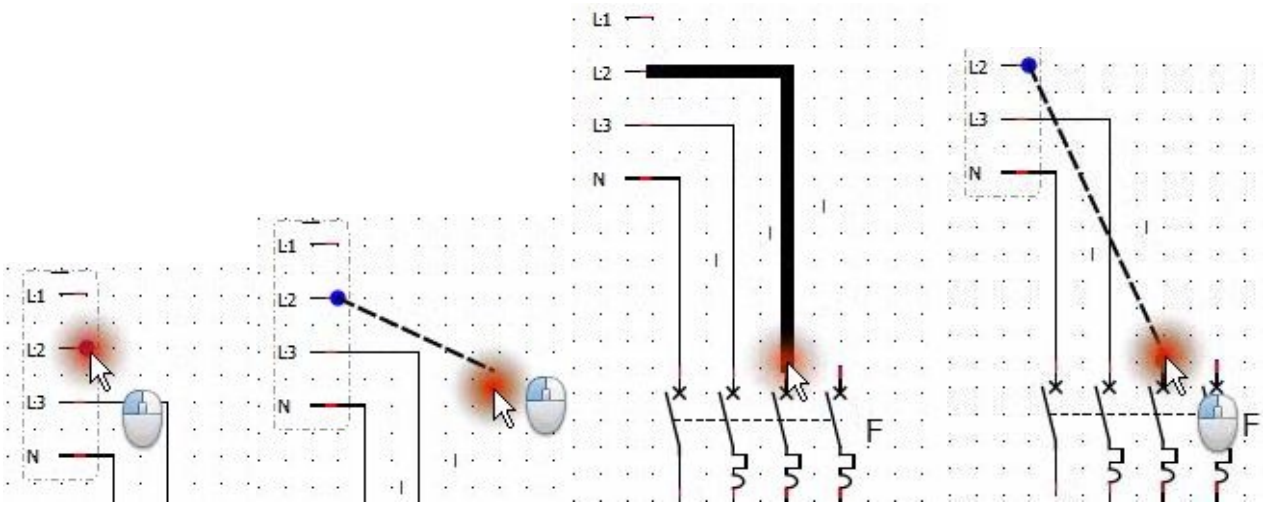
3. Release the left mouse button to validate



FTo manually link L2 with the circuit-breaker perform the following steps:

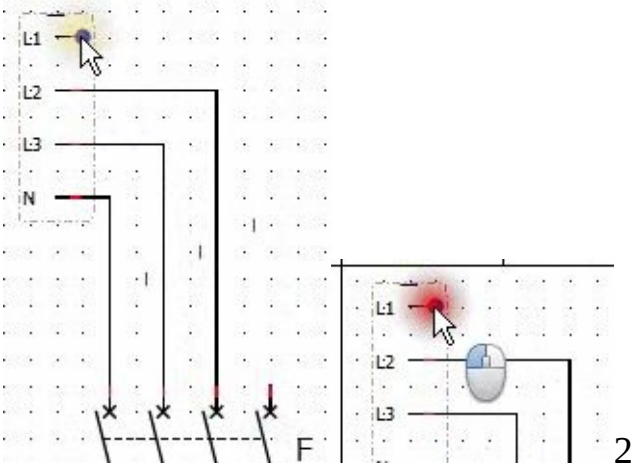
1. Move the mouse cursor until you select the end of the L2 terminal, Click and hold the left mouse button to maintain the selection

2. Move the cursor to the end of the 3rd contact of the Circuit-breaker
3. Release the left mouse button to validate

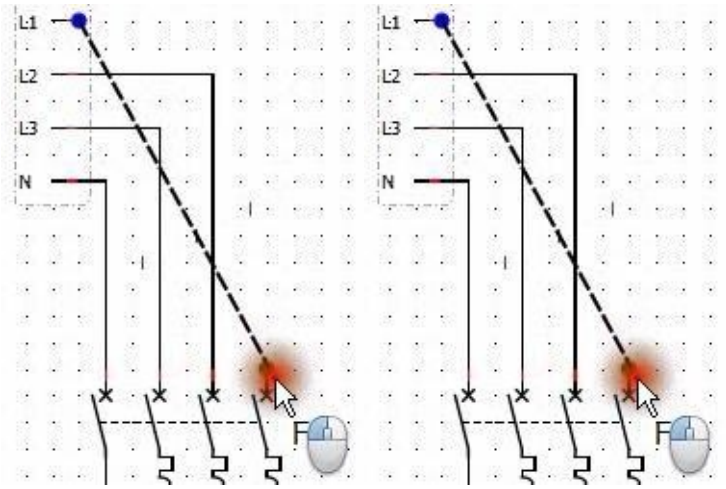


Finally manually link L1 to the Circuit-breaker by performing the following steps to complete the link between the source and the circuit-breaker:

1. Move the mouse cursor until you select the end of the L1 terminal, Click and hold the left mouse button to maintain the selection



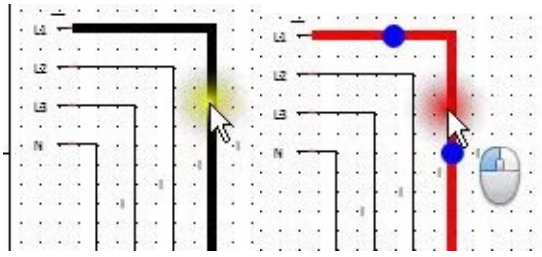
2. Move the cursor to the end of the last contact of the Circuit-breaker
3. Release the mouse button to validate



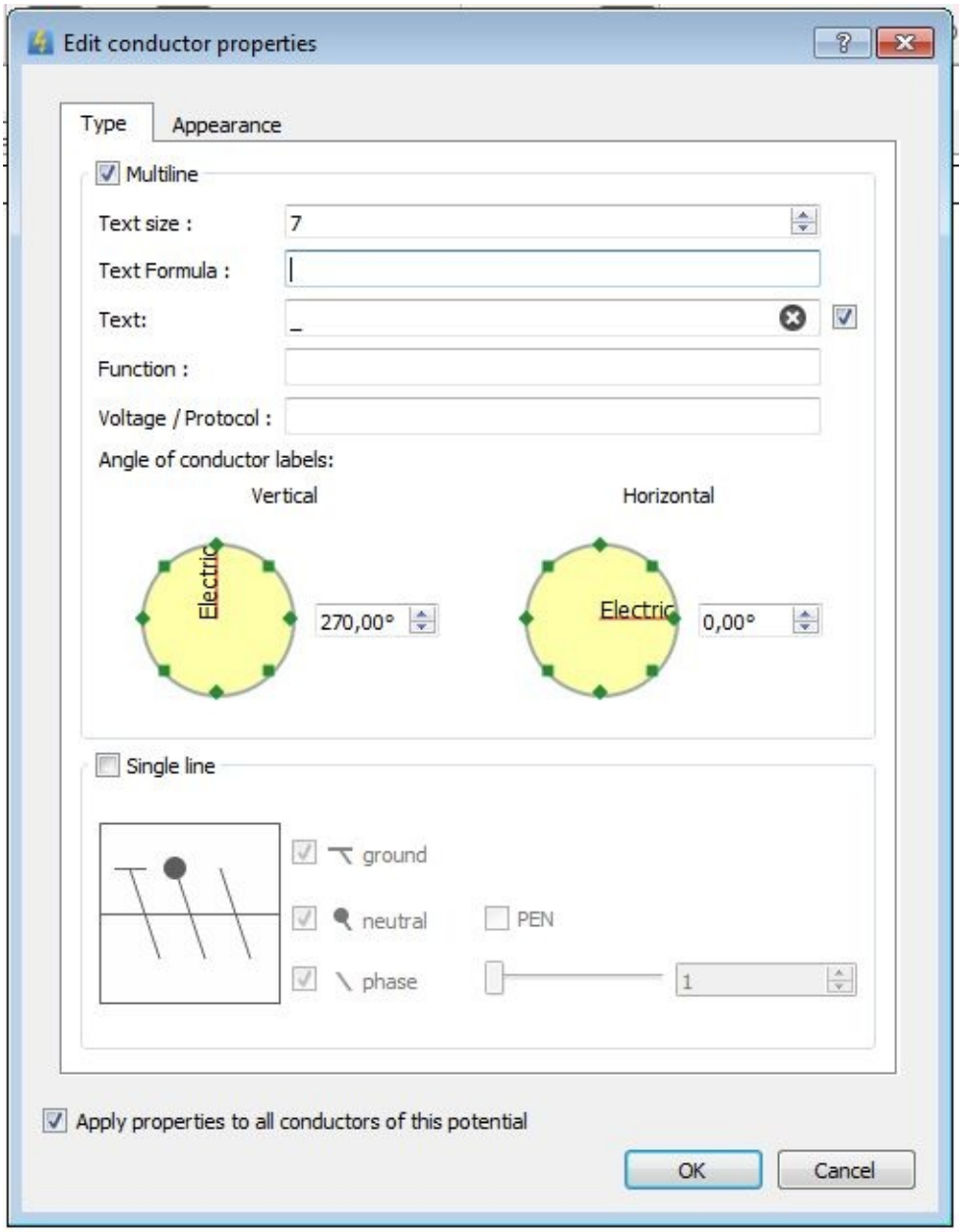
2.14 Modify Attributes of elements:

2.14.1 Changing the name of a wire (a link): *2.14.1.1 Change the name of the wire between L1 and the Circuit-breaker:*

1. Move the cursor until you select the connecting wire between L1 and the Circuit-breaker
2. double click with the left mouse button

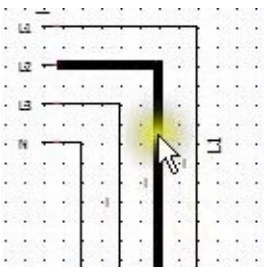

double click

3. Wait for the conductor properties editor to appear.
4. Clear the contents of the text box "Text" and type 'L1'
5. Validate the change by clicking OK

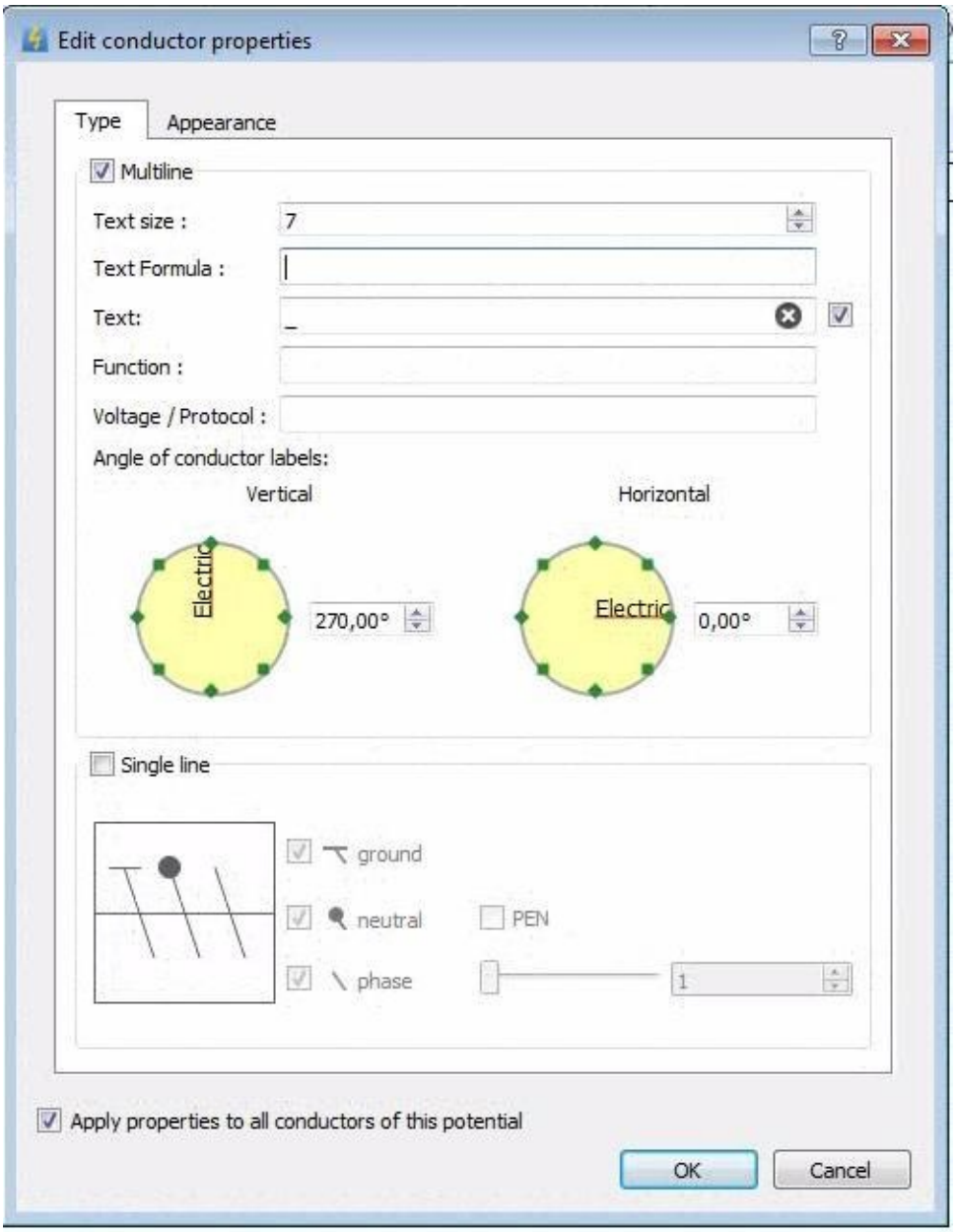


2.14.1.2Change the name of the wire between L2 and the Circuit-breaker:

1. Move the cursor until you select the connecting wire between L2 and the Circuit-breaker

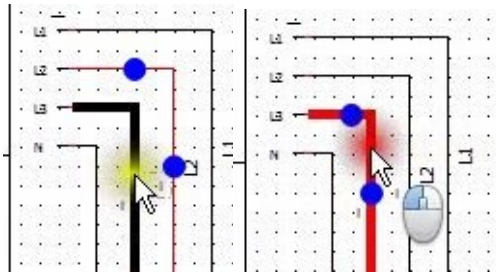


3. Wait for the conductor property editor to appear.
4. Clear the contents of the text box "Text" and type 'L2'
5. Validate the change by clicking OK

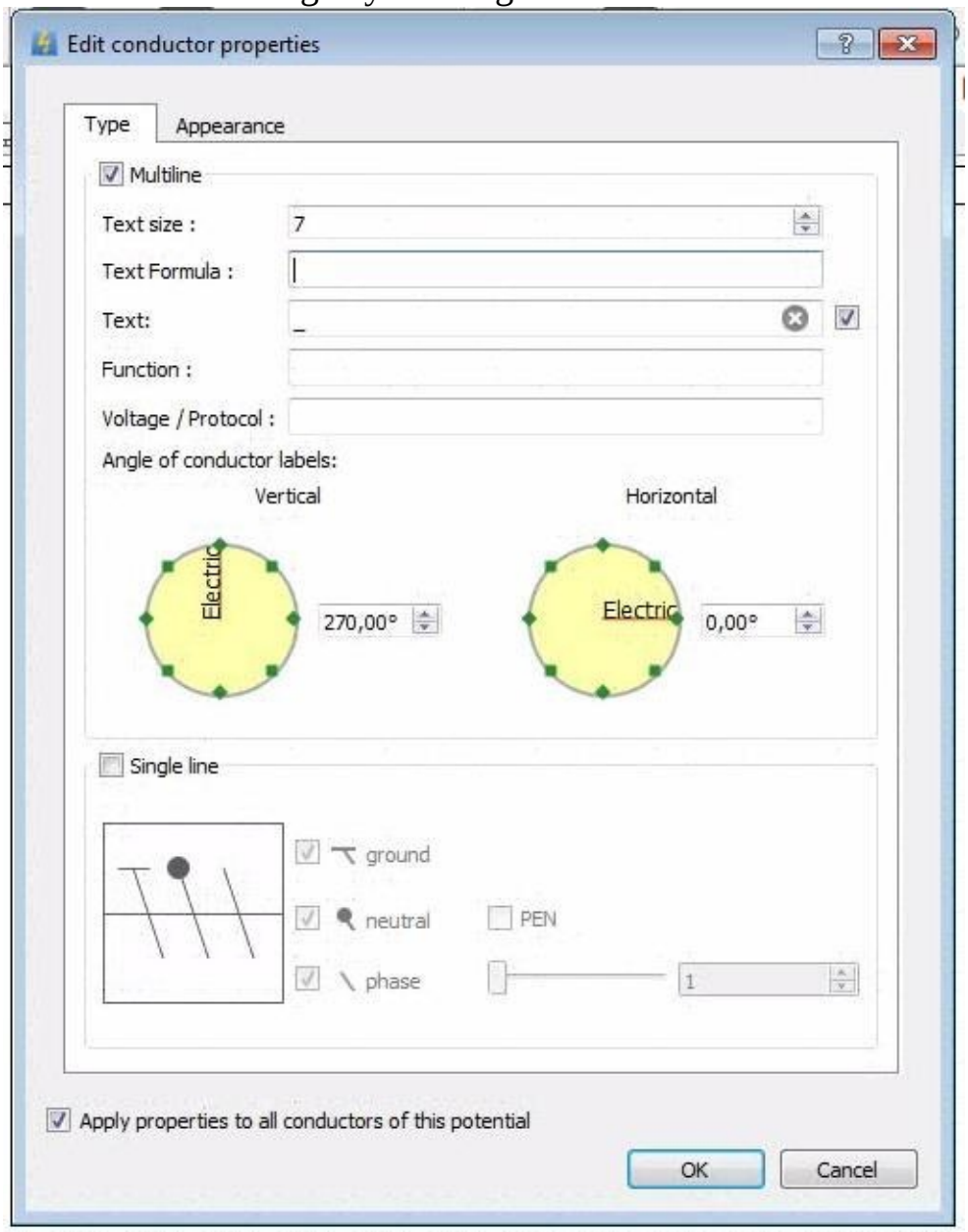


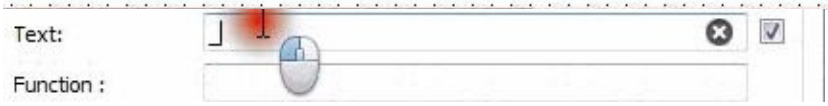
2.14.1.3Change the name of the wire between L3 and the Circuit-breaker:

1. Move the cursor until you select the connecting wire between L3 and the Circuitbreaker

2. double click with the left mouse button  3. Wait for the conductor property editor to appear.

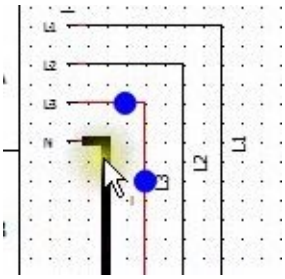
4. Delete the contents of the "Text" text box and type 'L3'
5. Validate the change by clicking OK





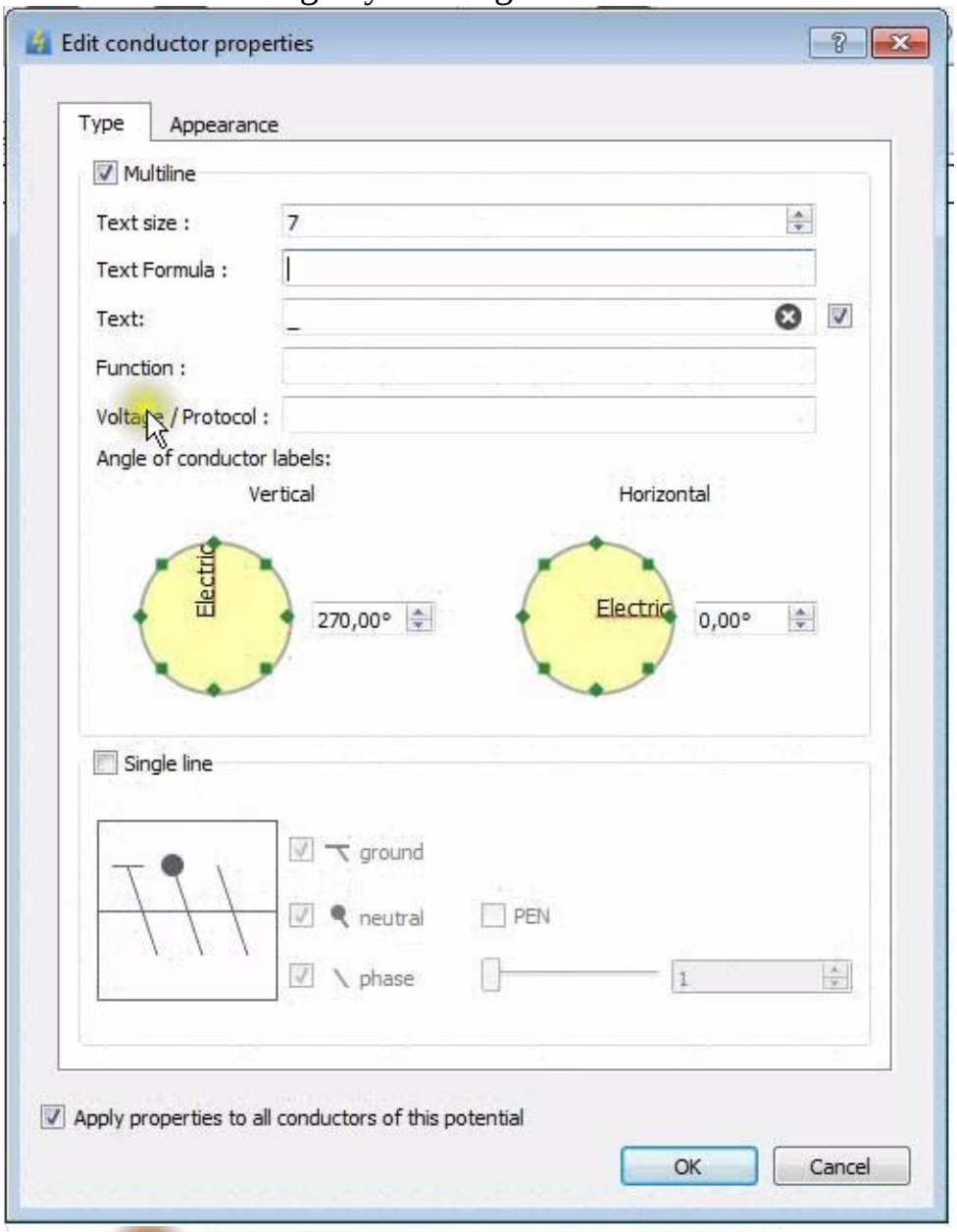
2.14.1.4Change the name of the wire between N and the Circuit-breaker:

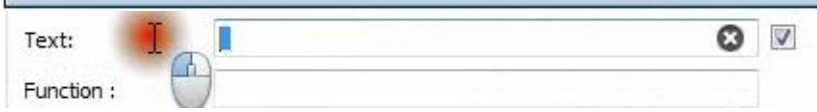
1. Move the cursor until you select the wire between N and the Circuit-breaker



3. Wait for the conductor property editor to appear.

4. Clear the contents of the "Text" text box and type 'N'
5. Validate the change by clicking OK

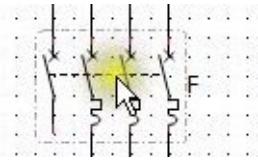




2.14.2 Changing

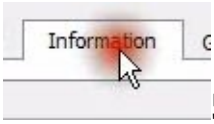
the Circuit-breaker name:

1. Move the cursor until you select the Circuit-breaker



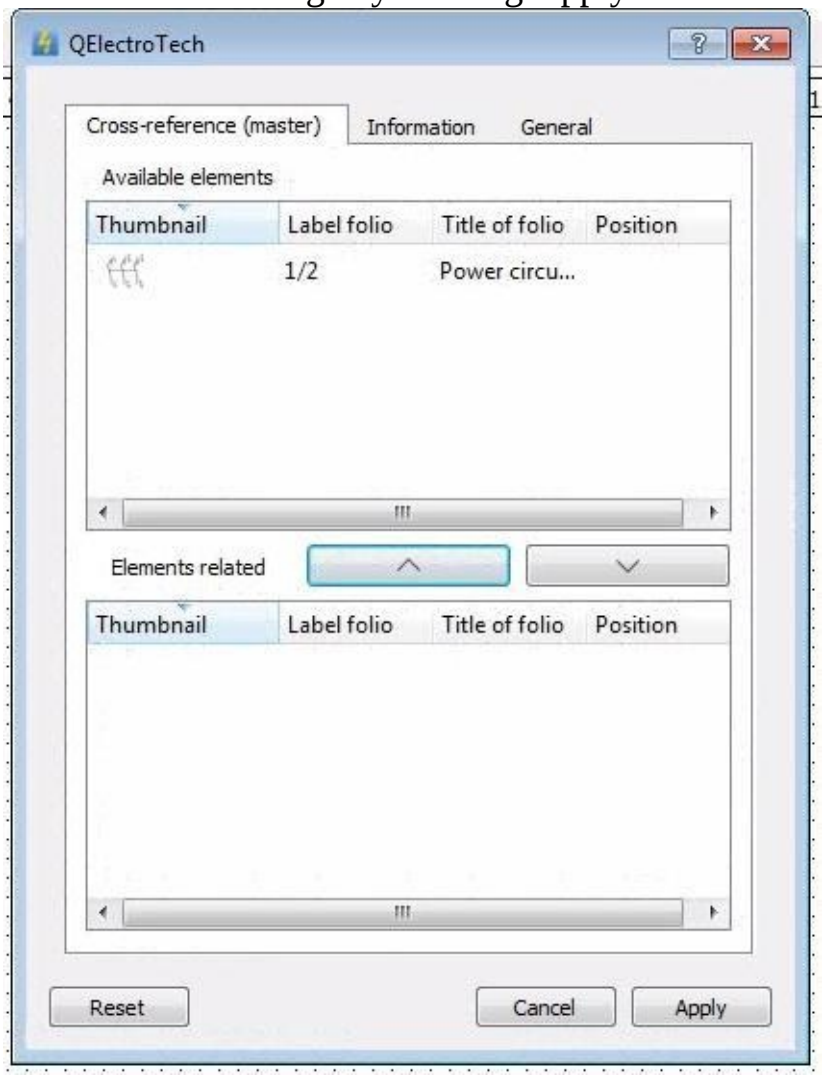
3. Wait for the element properties editor to appear.

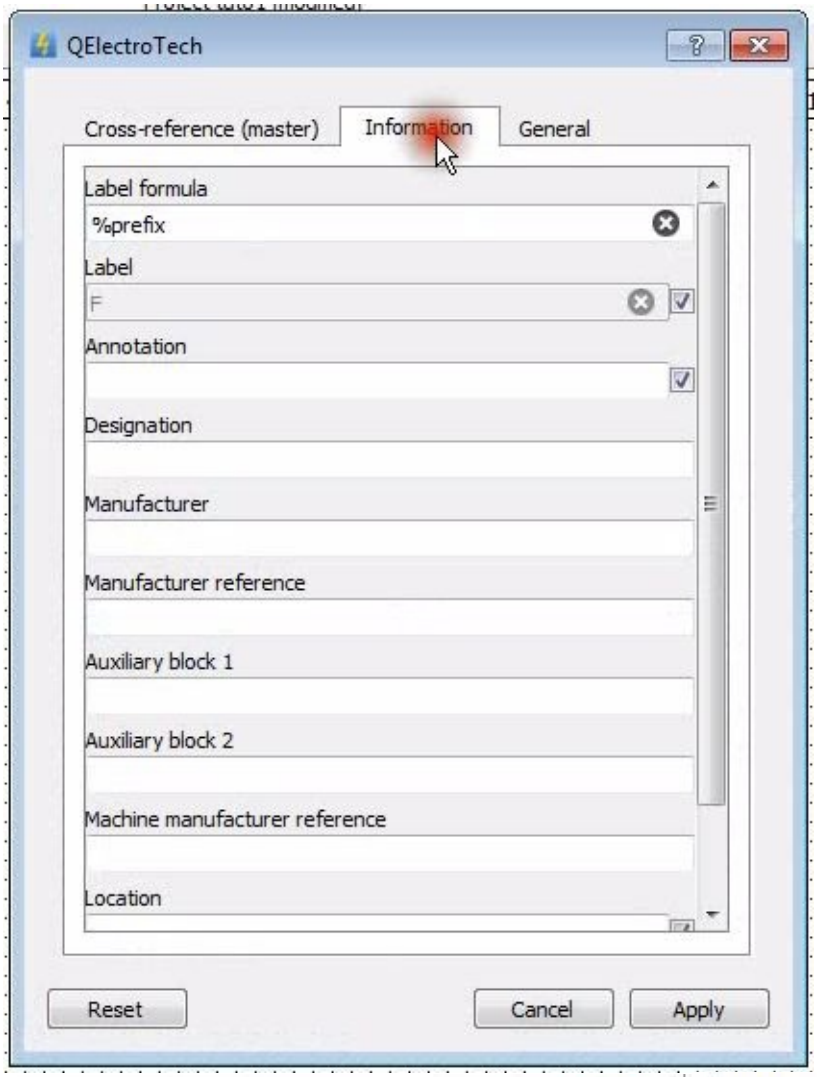
4. click on Information



5. disable the Label formula and clear the contents of the Label box and type F1

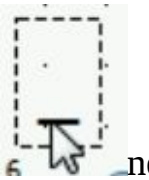
6. Validate the change by clicking Apply



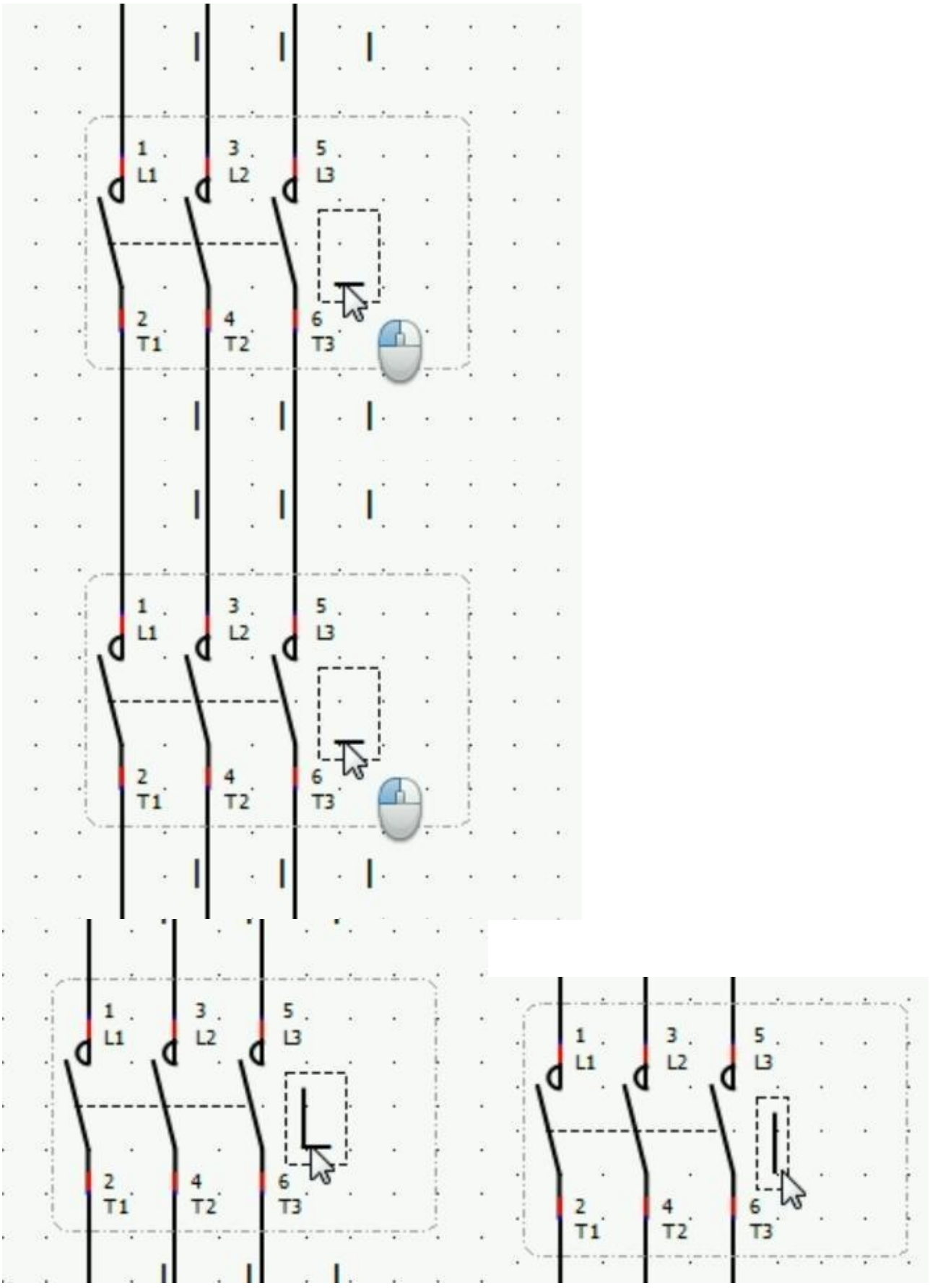


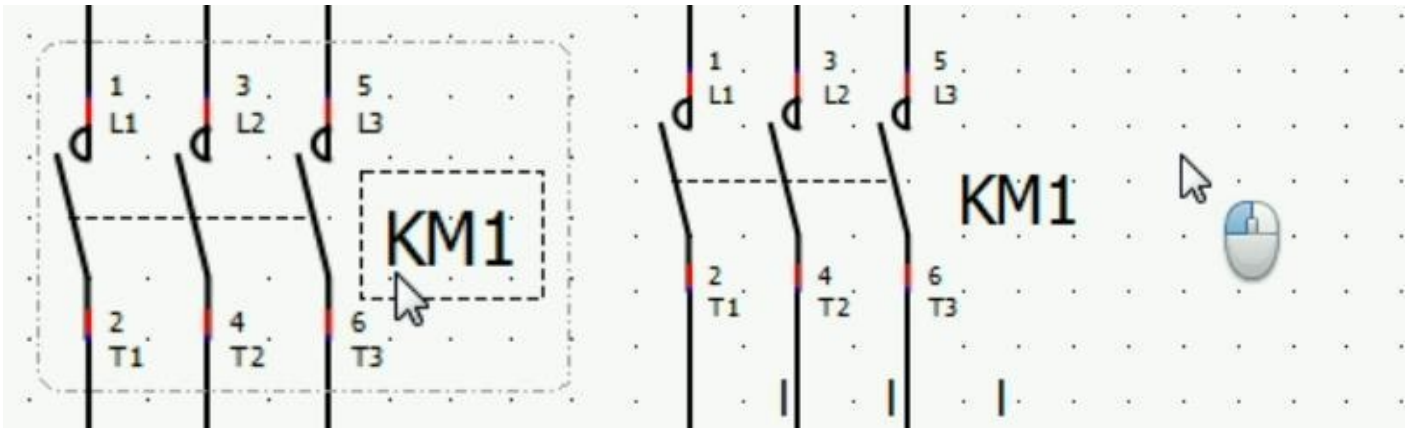
2.14.3 Changing the name of the contactor:

1. Move the cursor until you select



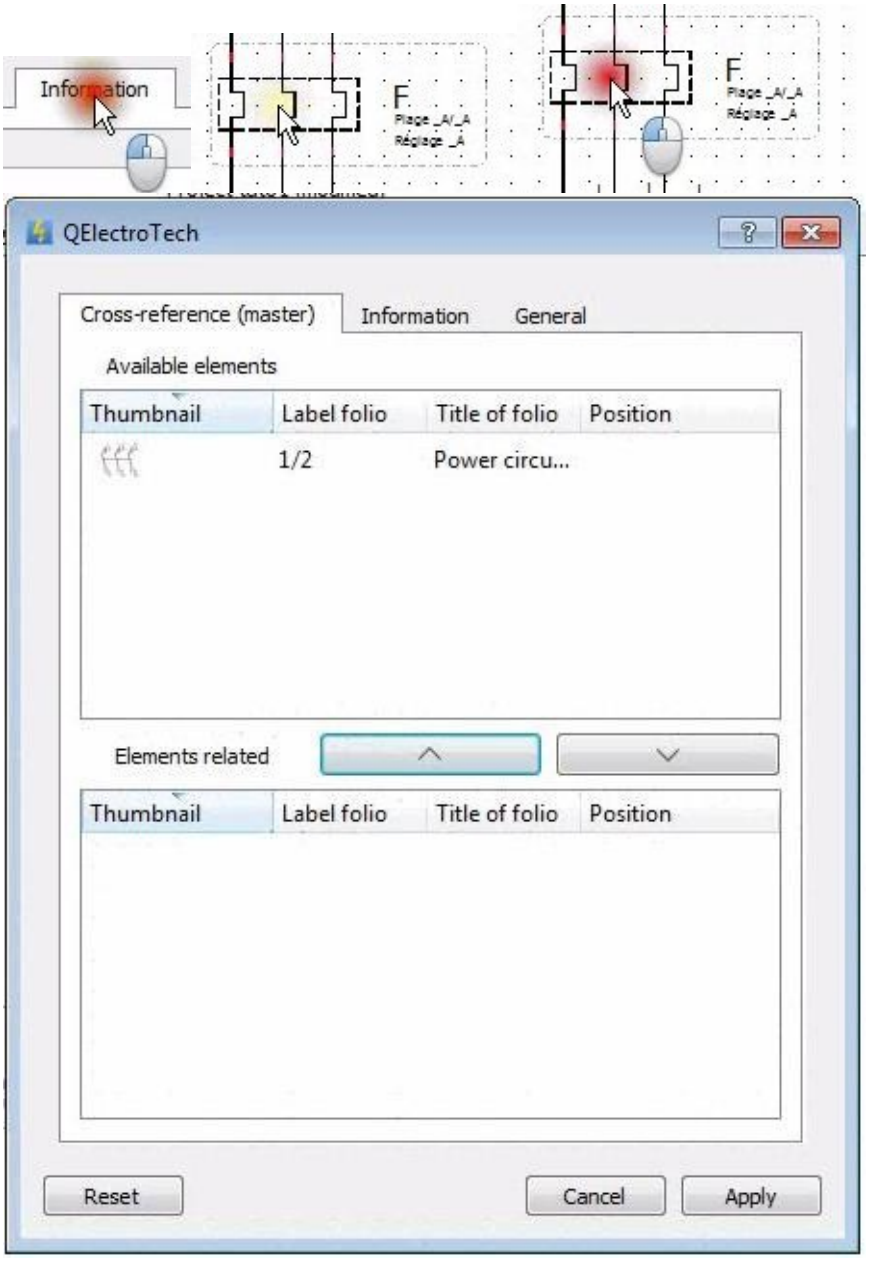
next to the Contact power contactor by clicking with the left mouse button
2. double click with the left mouse button to edit the contents of this box
3. Wait for the cursor to start flashing in this box.
4. Clear the contents with the 'Delete' button on the keyboard and type 'KM1'
5. Validate the change by clicking on the drawing sheet away from this box

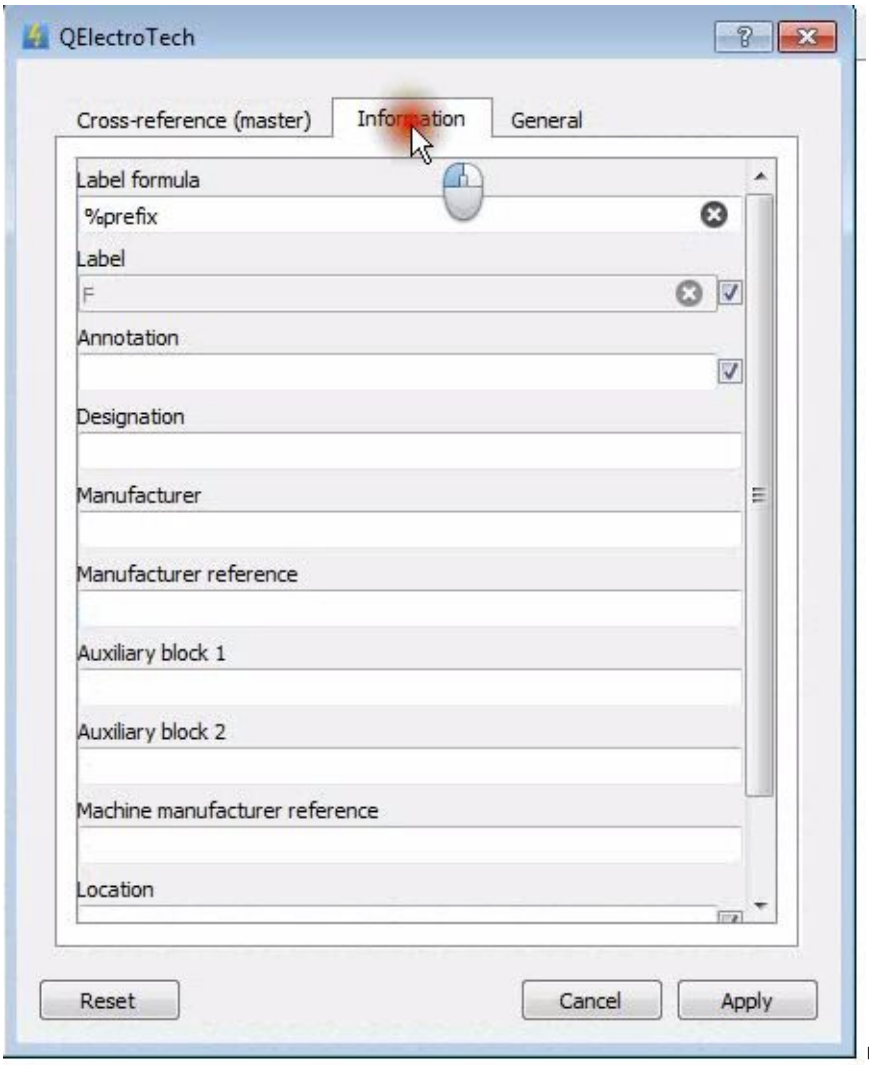

1
L1
3
L2
5
L3
2
T1
4
T2
6
T3
1
L1
3
L2
5
L3
2
T1
4
T2
6
T3
1
L1
3
L2
5
L3
2
T1
4
T2
6
T3
1
L1
3
L2
5
L3
2
T1
4
T2
6
T3



2.14.4 Change the name of the Thermal Relay:

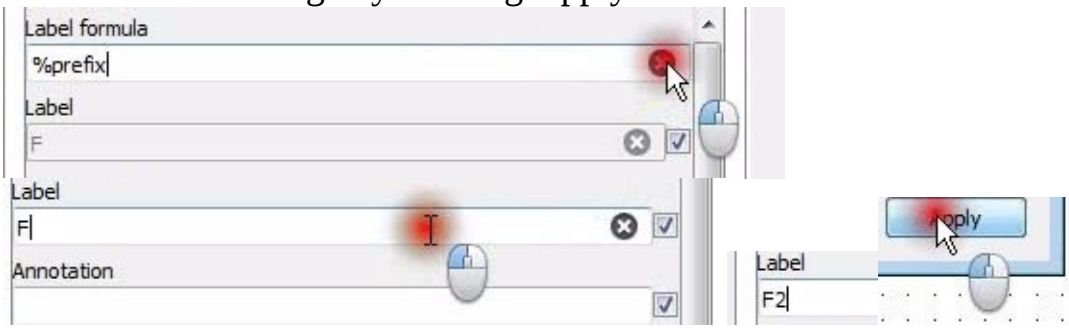
1. Move the cursor until you select the Thermal Relay
2. double-click with the left mouse button
3. Wait for the element properties editor to appear.
4. click on Information





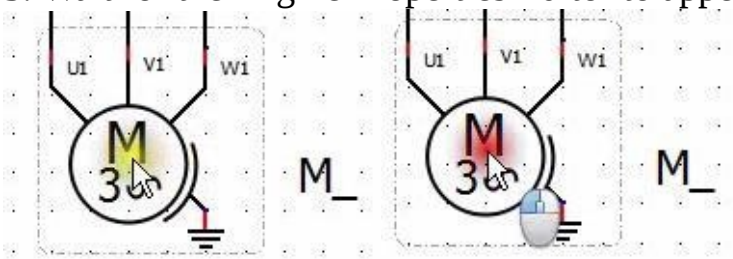
5. disable the Label formula and clear the contents of the Label box and type F2

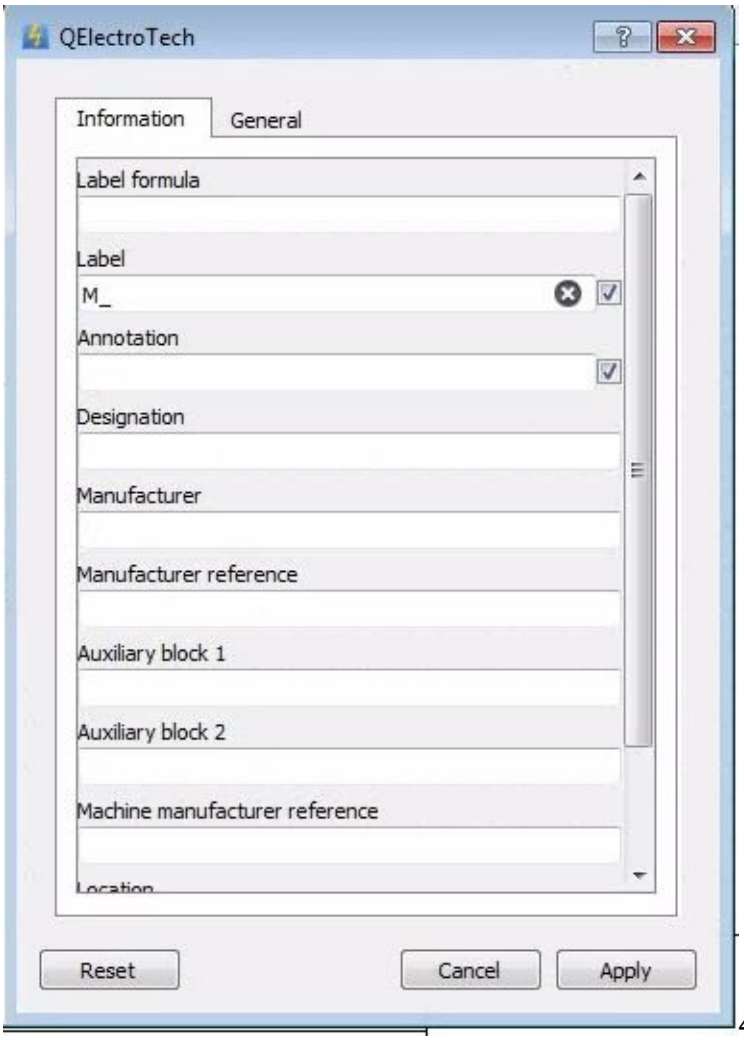
5. Validate the change by clicking Apply



2.14.5 Change the name of the Engine:

1. Move the cursor until you select the Engine
2. Double click with the left mouse button.
3. Wait for the Engine Properties Editor to appear.





4. Clear the contents of the "Label" text box and type 'M1'

5. Validate the change by clicking Apply

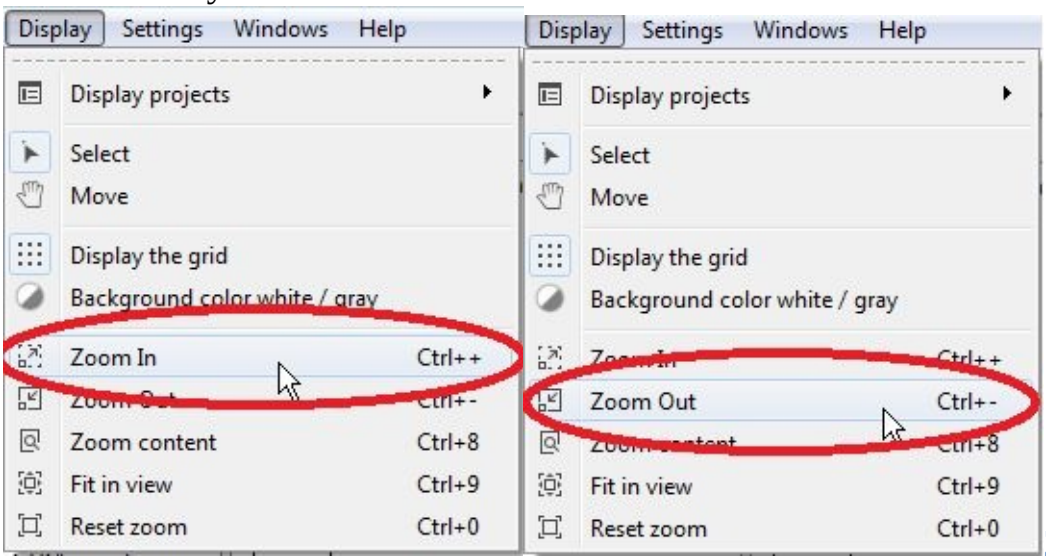
2.15 The tips: 2.15.1 Zoom:

To zoom in: go to Display menu ✉ Zoom In

Or use the keys: "Ctrl" with "+"

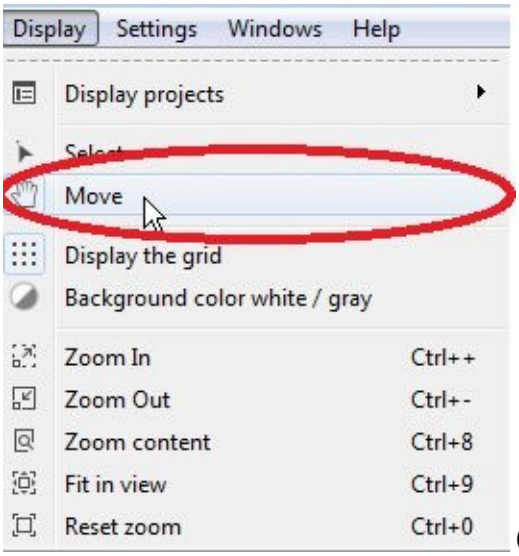
To zoom out: go to Display menu✉ Zoom Out

Or use the keys: « Ctrl » with « - »

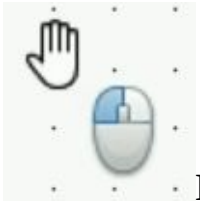


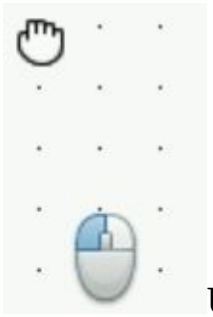
2.15.2 View mode:

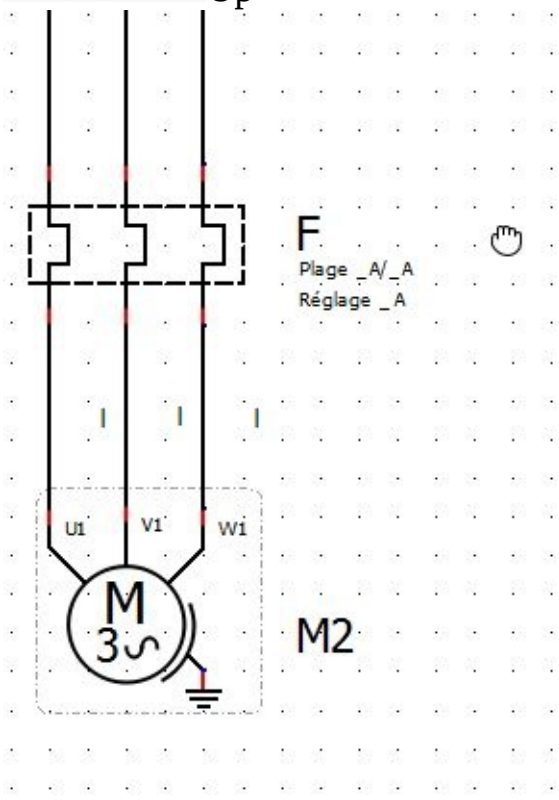
After zooming in, probably, you would need to move the drawing sheet to see the rest and that's where you'll need the Move Mode; To activate it go to the menu Display✉Move



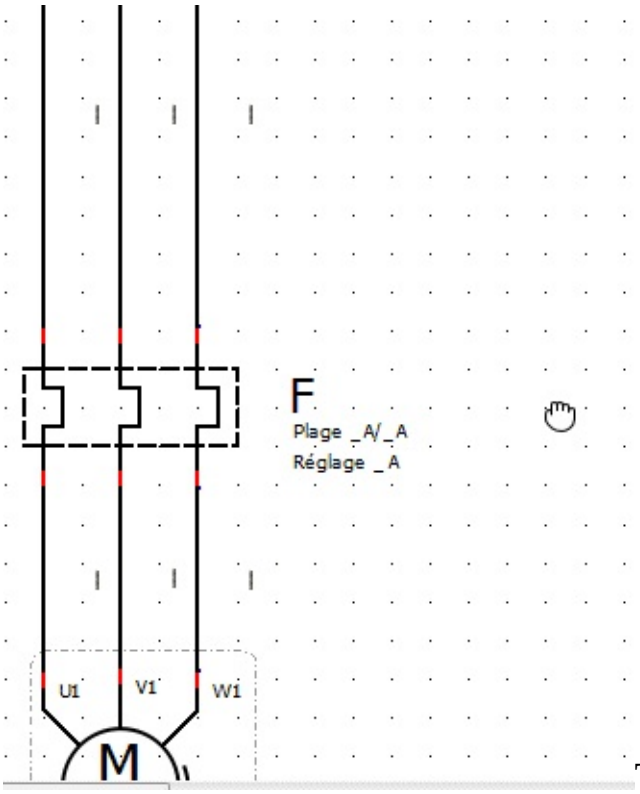
Click on the left mouse button


Hold the button and move the mouse to move the sheet

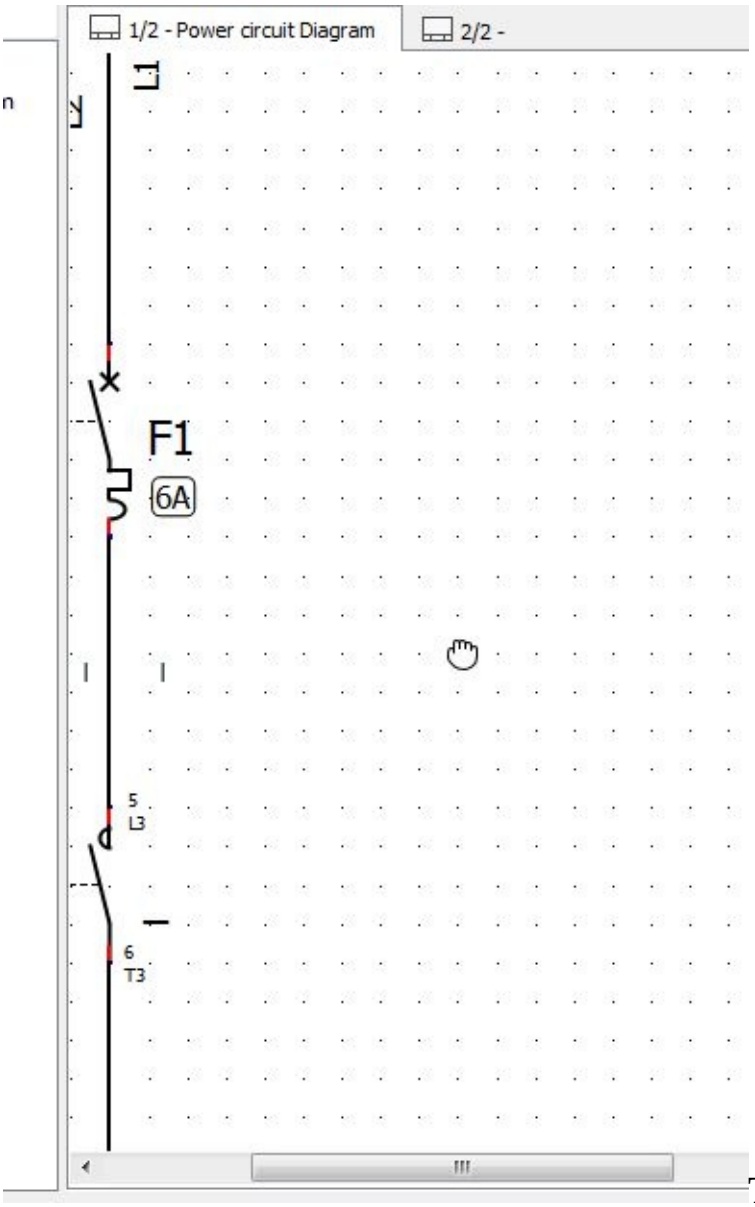

Up



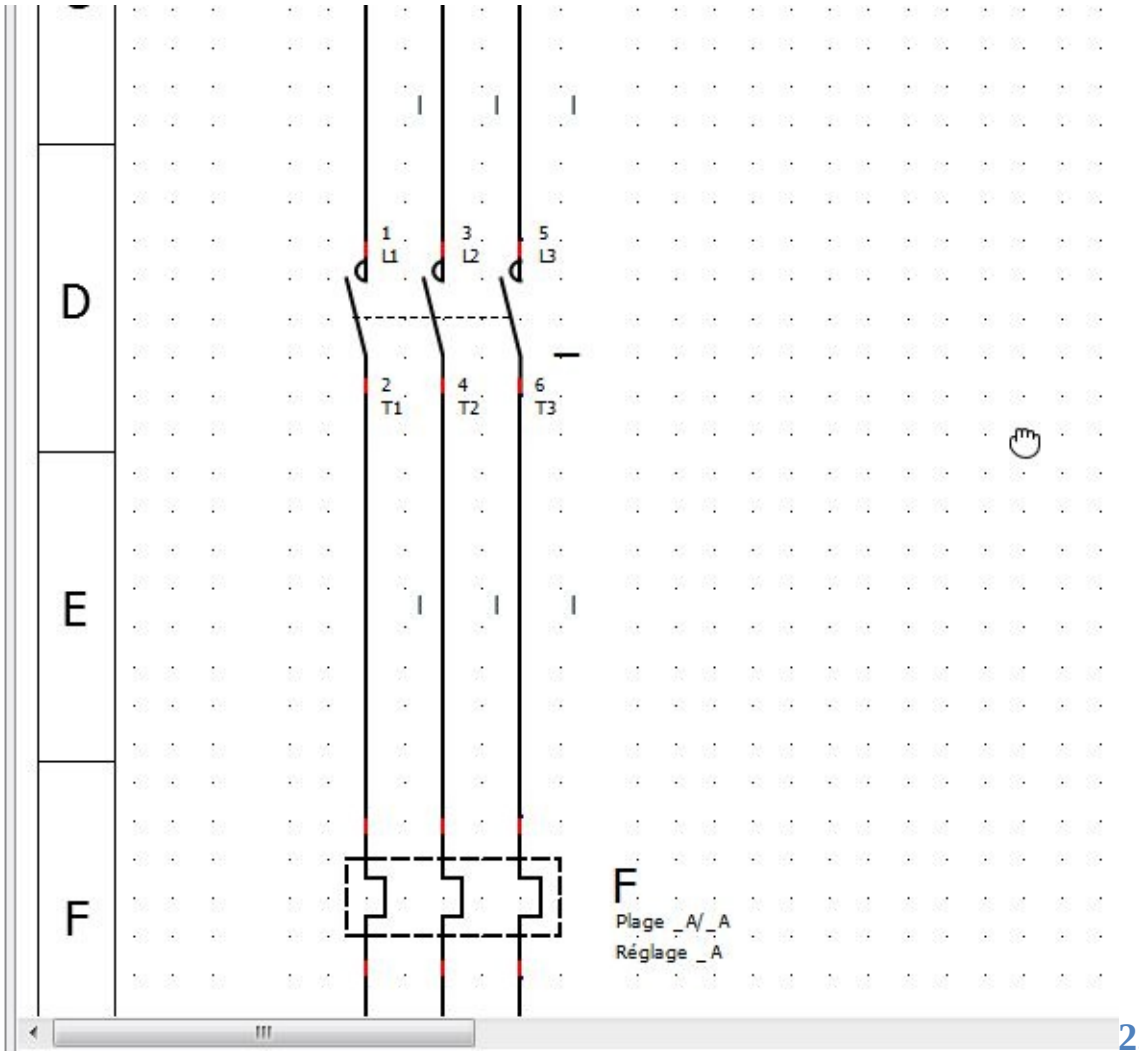
Down



To the left



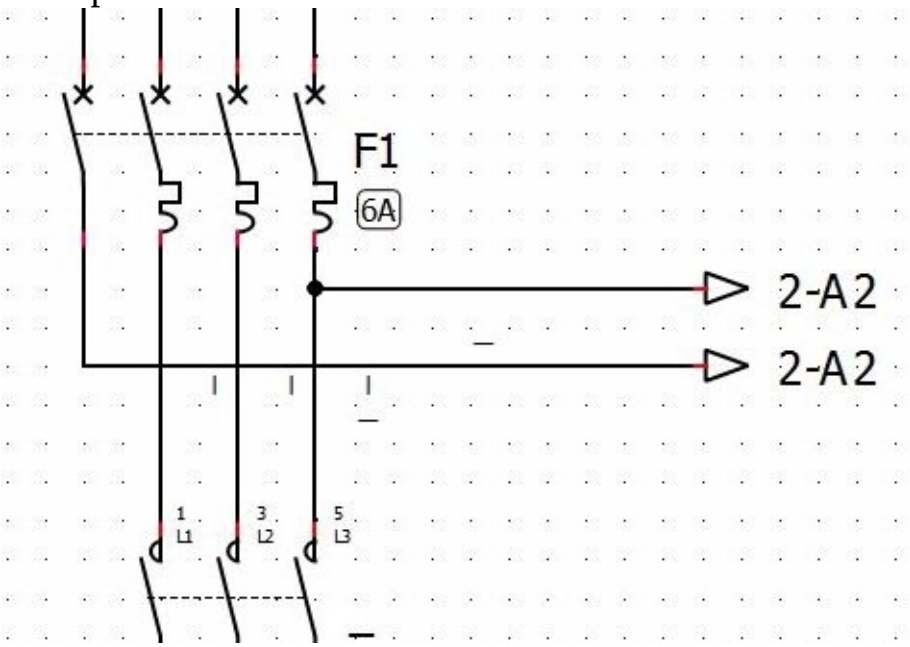
To the right



2.15.3 Folio à Folio:

Now to see how to link the diagrams of the two sheets we will add a control diagram to the power diagram already realized in the example of section 1.12 and link them by the "folio" jumps.

Example:“folio” to“folio”:

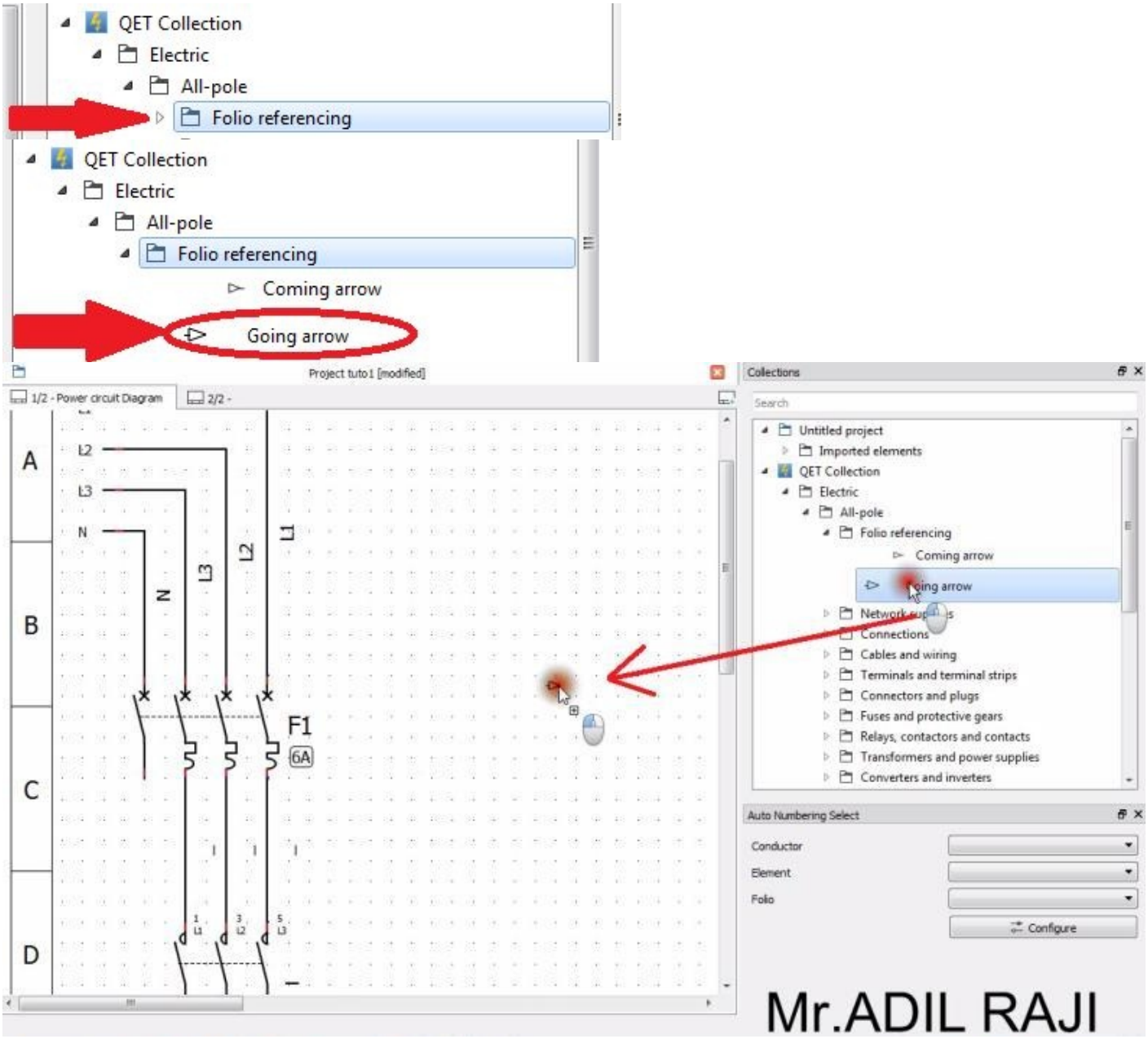


To have the same result as in the example above, we will perform the following steps:

go to QET Collection 🗹 Electric 🗹 All-pole 🗹 Folio referencing

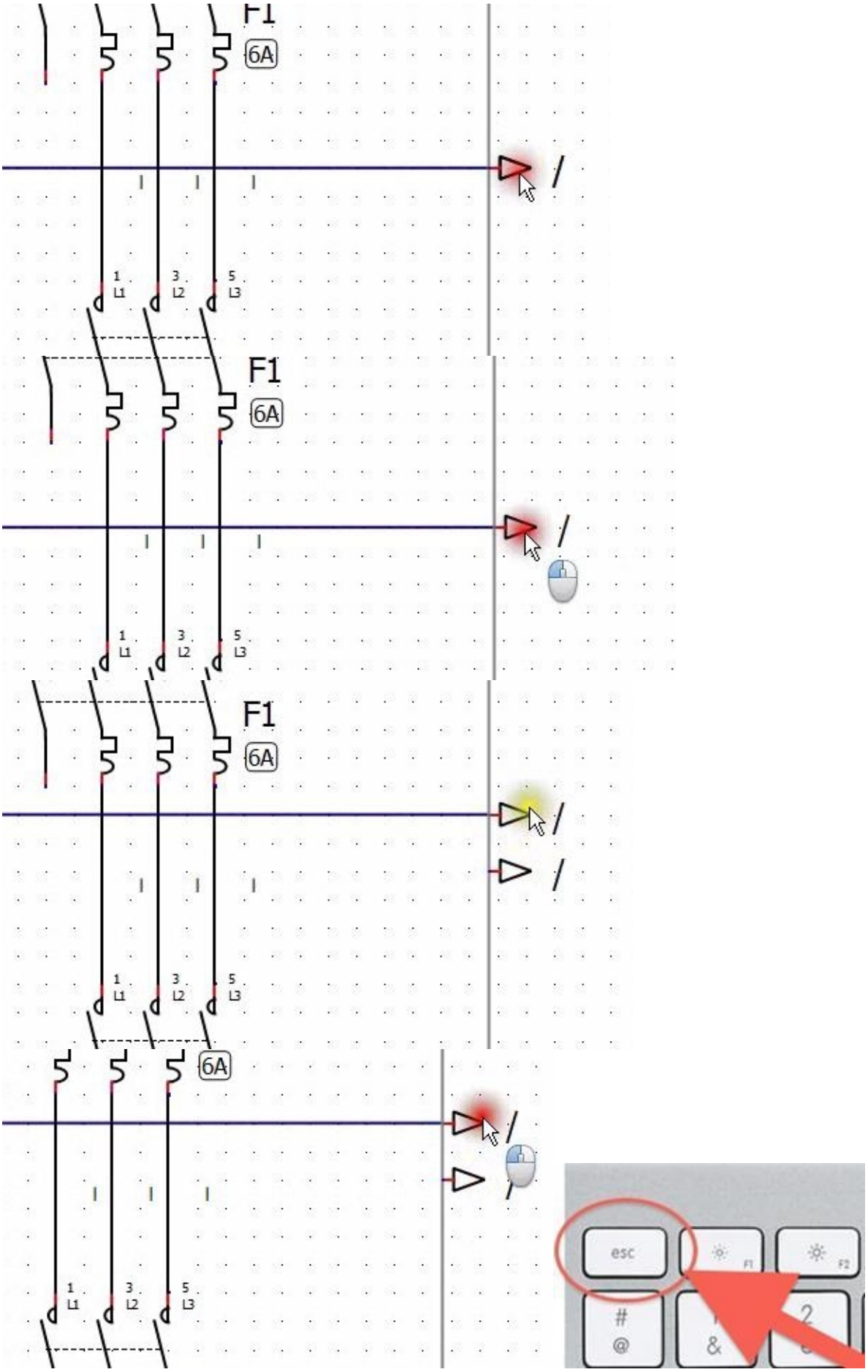
Select : Going arrow

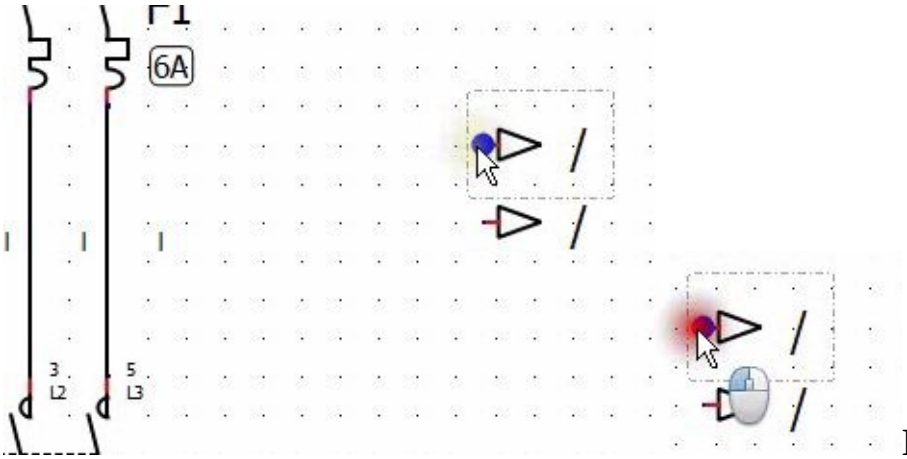
drag / drop



Move the cursor to place the selected item between the breaker and the contactor, on

the right
Click to confirm location
Move the cursor another time
Click to confirm location
Click on esc button
Move the cursor until you select the end of the first "going arrow" element
click and hold the left mouse button to maintain the selection

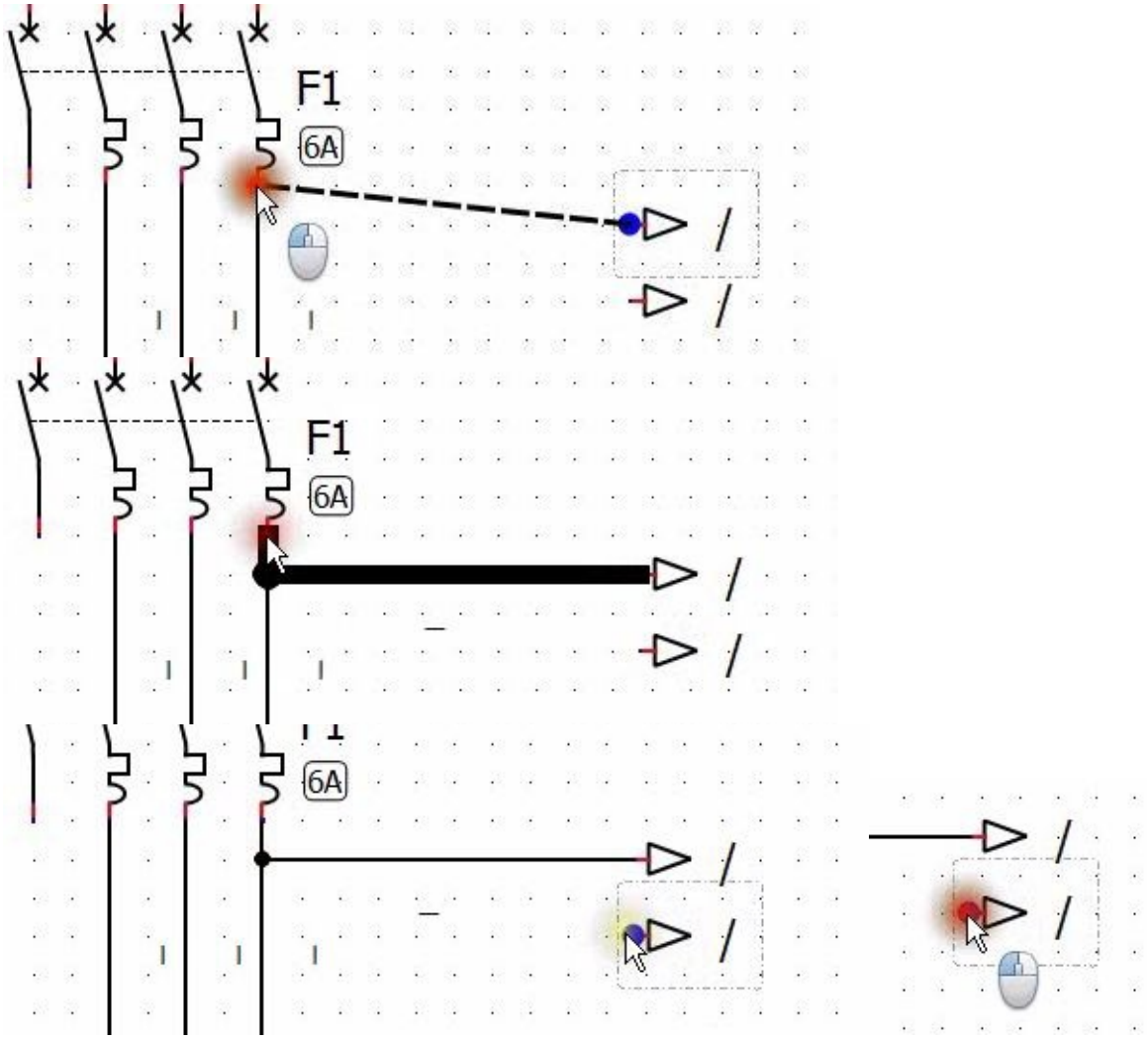

F1
6A
1
L1
3
L2
5
L3
F1
6A
1
L1
3
L2
5
L3
F1
6A
1
L1
3
L2
5
L3
6A
1
L1
3
L2
5
L3
esc
F1
F2
#
@
&
2

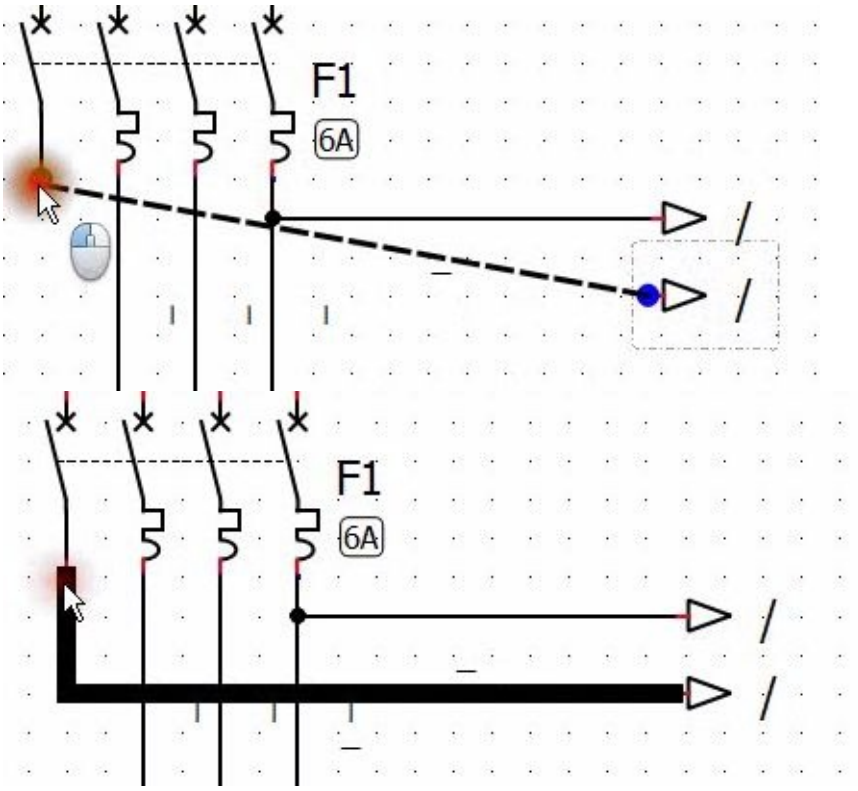


Move the cursor to the right end of the circuit-breaker...
Release the left mouse button to validate

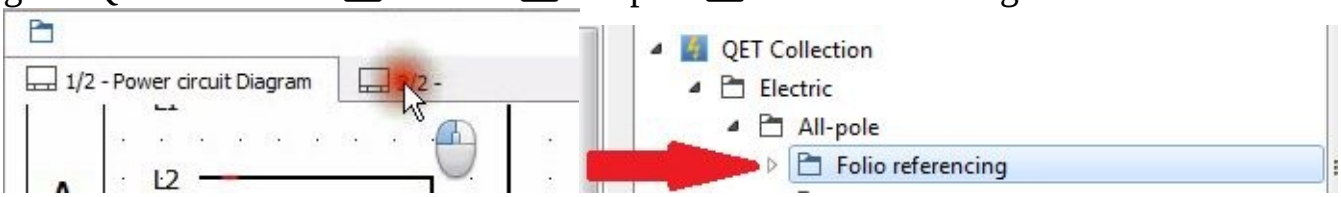
Move the mouse cursor until you select the end of the second "going arrow" element

click and hold the left mouse button to maintain the selection
Move the cursor to the left end of the circuit-breaker
Release the left mouse button to validate

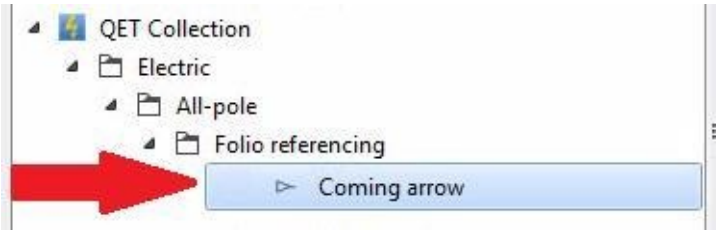


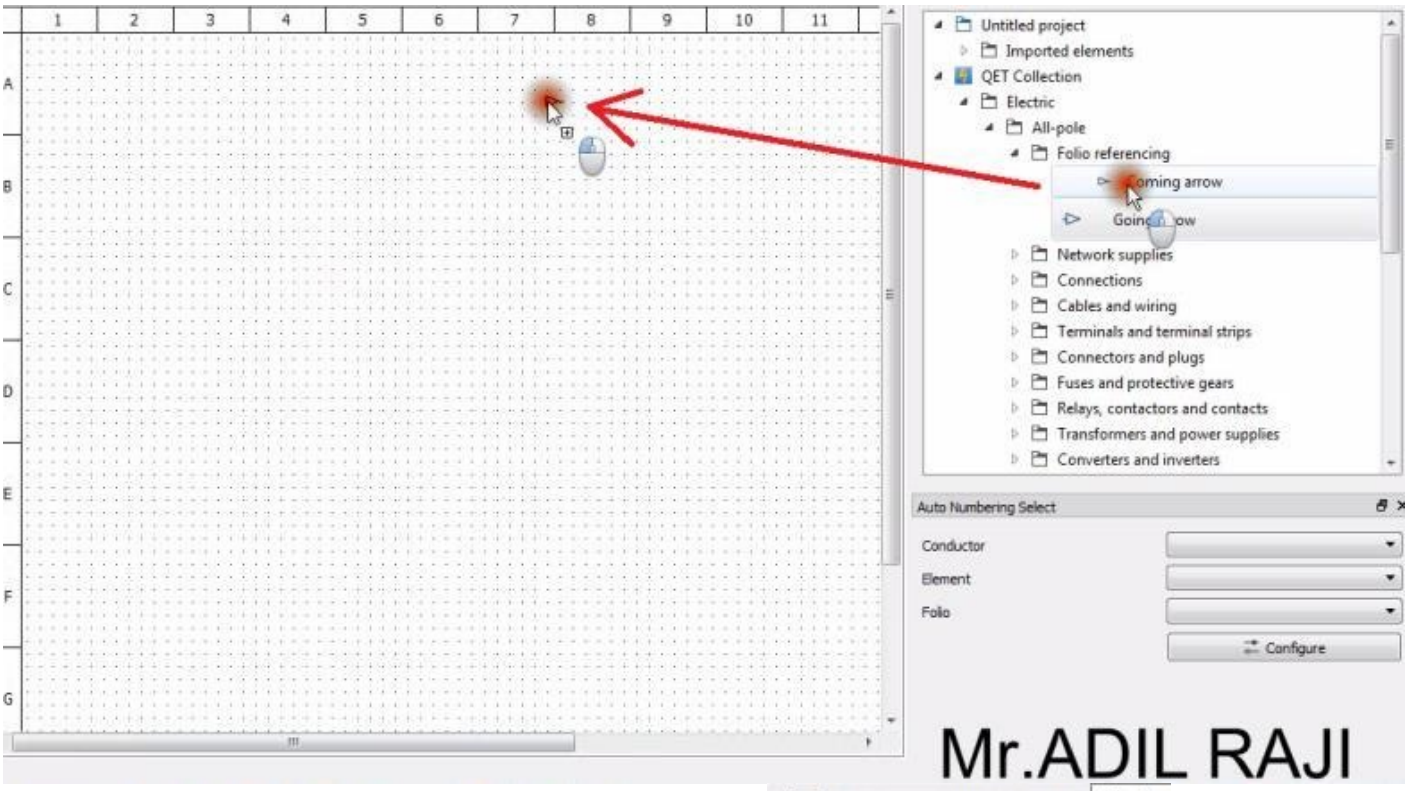


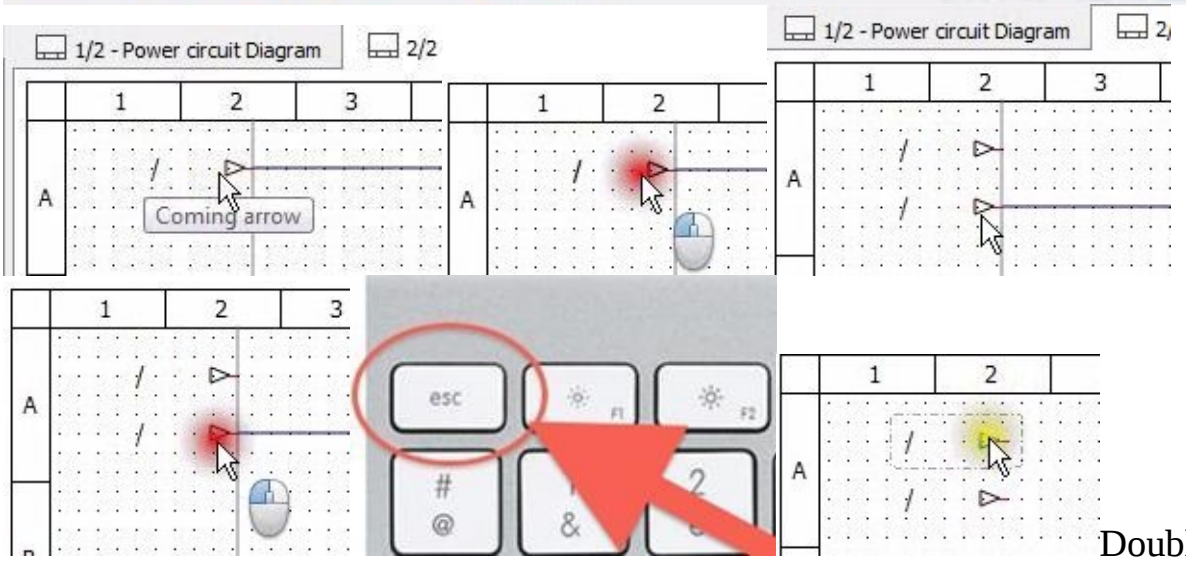
Now we will add the "Coming arrow" and link elements in the second sheet of the drawing.
Click on the second drawing sheet that we created before.
go to QET Collection ✉ Electric ✉ All-pole ✉ Folio referencing



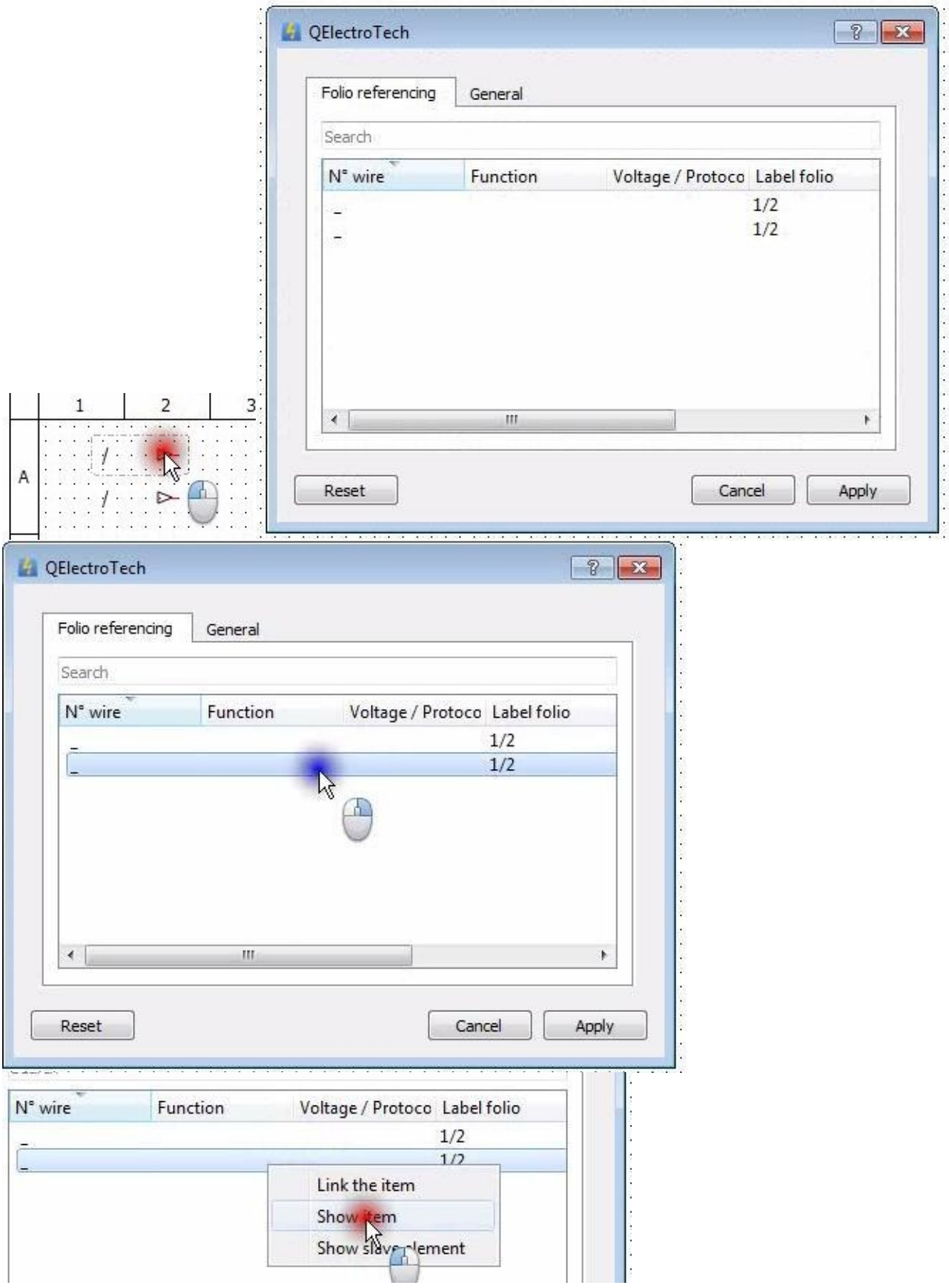
Select : Coming arrow
drag / drop
Move the cursor to place the selected item
Click to confirm location
Move the cursor another time
Click to confirm location
Click on esc button

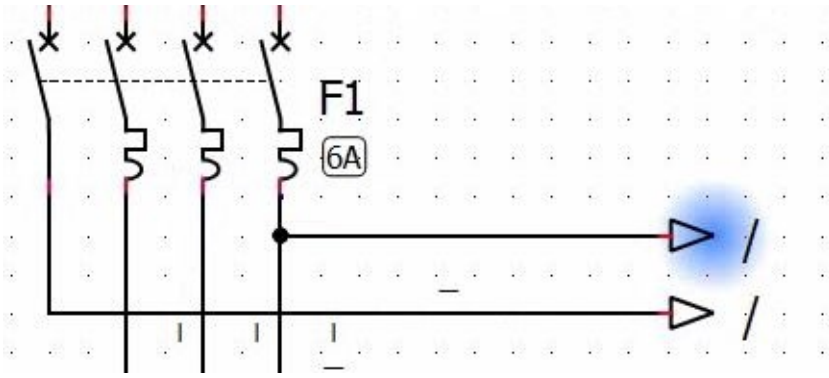
Select the one Coming arrow 





Double click with the left button.
Wait for the Properties Editor to appear.
Right click
Click on “Show item”
QElectrotech will show the folio of the selected line and highlight the arrow.


QElectroTech
Folio referencing
General
Search
N° wire
Function
Voltage / Protoco
Label folio
1/2
1/2
Reset
Cancel
Apply
1
2
3
A
QElectroTech
Folio referencing
General
Search
N° wire
Function
Voltage / Protoco
Label folio
1/2
1/2
Reset
Cancel
Apply
N° wire
Function
Voltage / Protoco
Label folio
1/2
Link the item
Show item
Show slave element

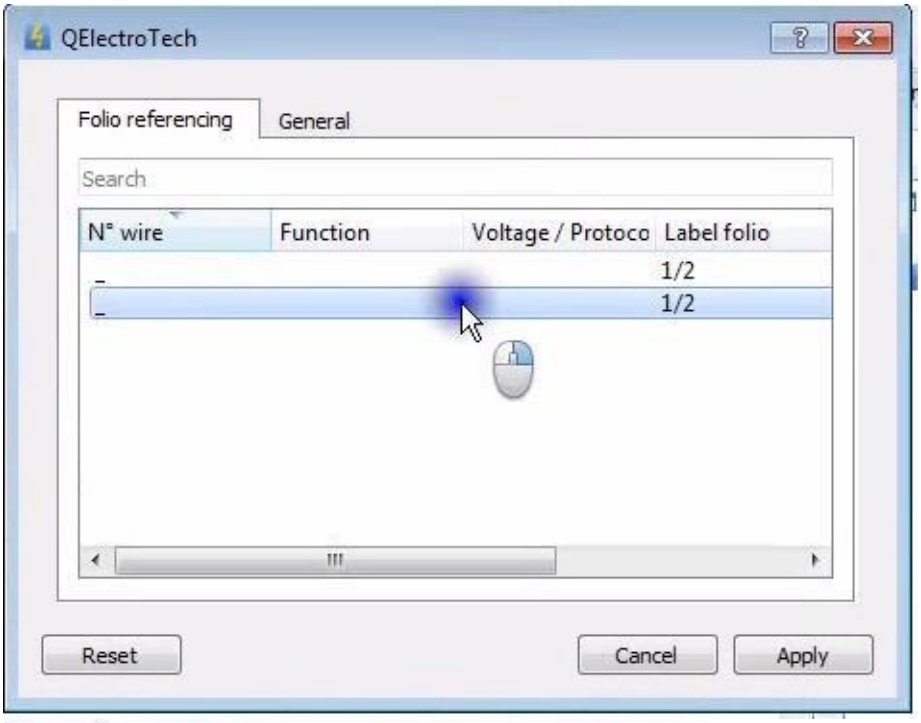


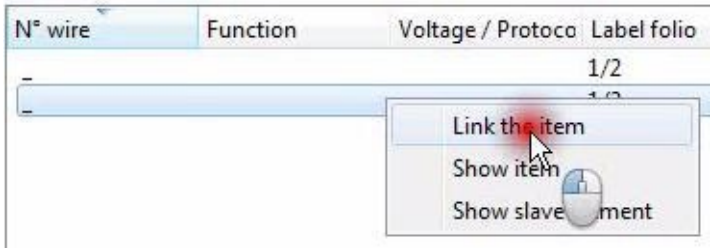
Right click again on the selected line
Click on “Link the item”
The color of the selected line will change to green

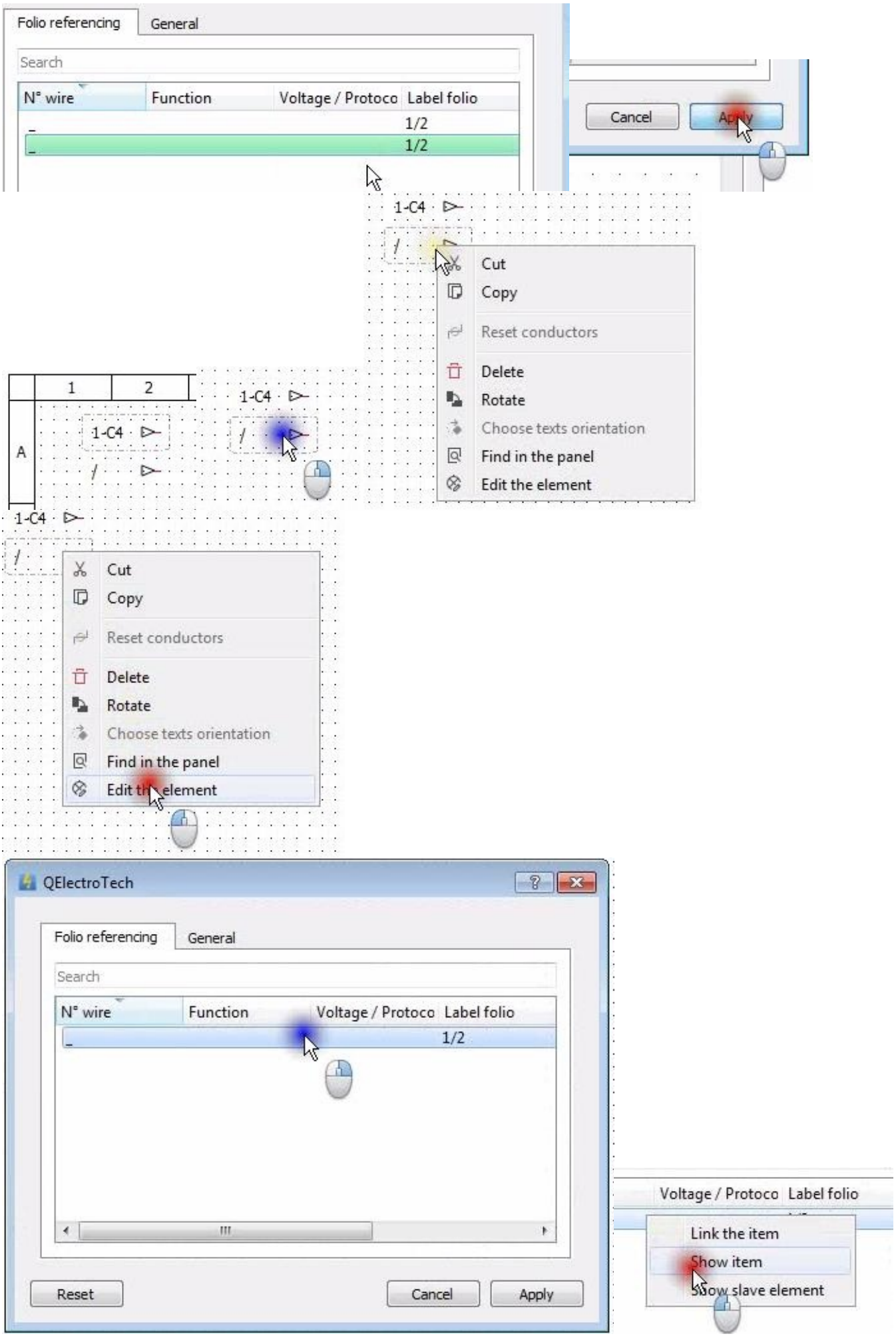
Click on Apply to validate the cross reference between the arrow of this diagram with the other present in the "power diagram"in “folio1” located in “C4”

Right click on the second “coming arrow” and click on Edit the element
Right click on the remaining line

And click on “Show item”






Folio referencing
General
Search
N° wire
Function
Voltage / Protoco
Label folio
1/2
1/2
Cancel
Apply
1-C4
Cut
Copy
Reset conductors
Delete
Rotate
Choose texts orientation
Find in the panel
Edit the element
1
2
A
QElectroTech
Reset
Cancel
Apply
Link the item
Show item
Show slave element

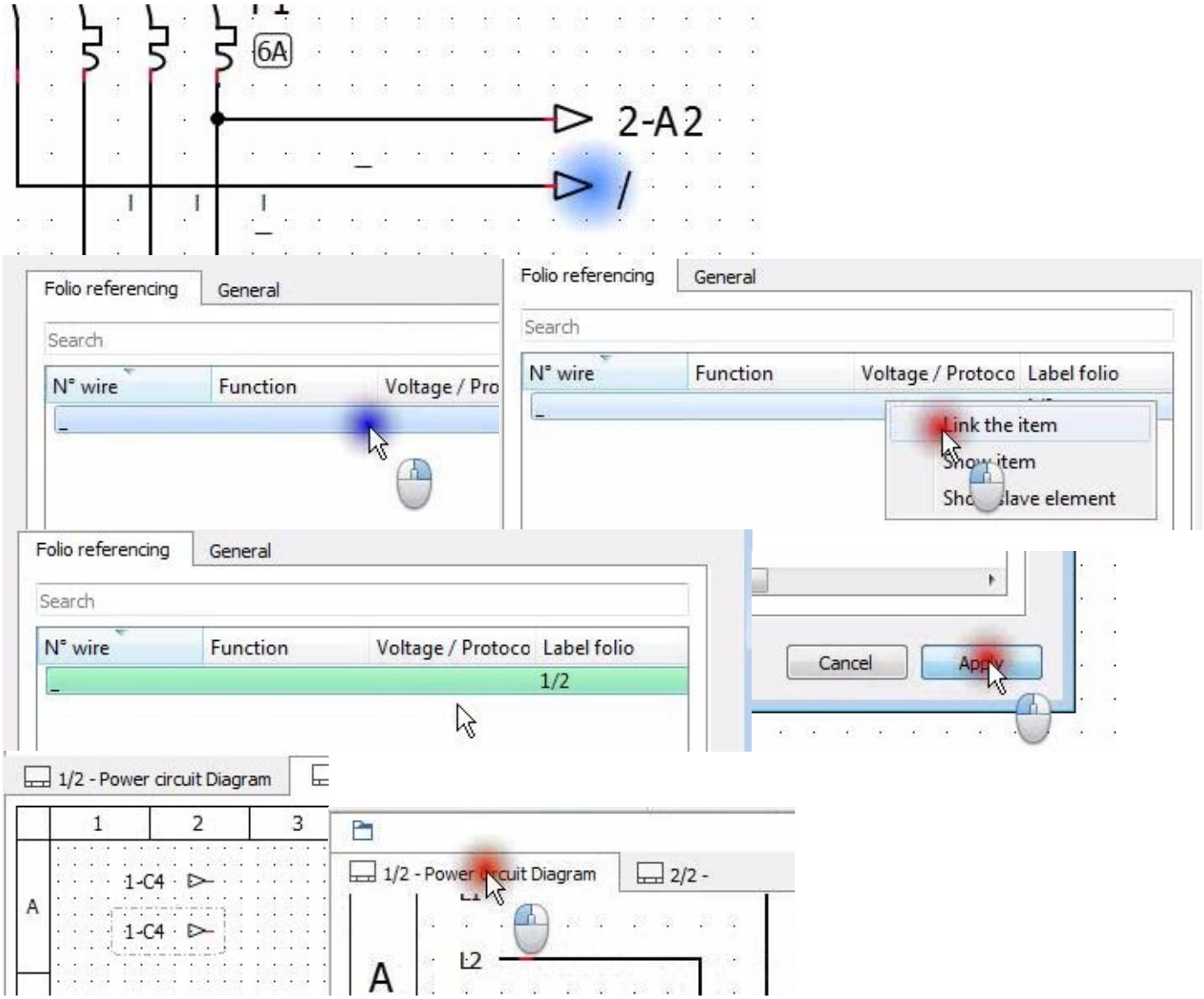
QElectrotech will show the folio of the selected line and highlight the arrow.

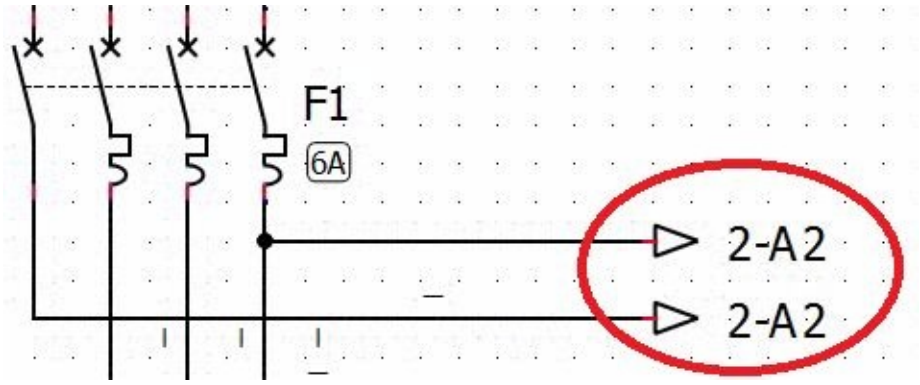
Right click again on the selected line
Click on “Link the item”
The color of the selected line will change to green

Click on Apply to validate the cross reference between the arrow of this diagram with the other present in the "power
diagram"in “folio1” located in “C4”

Click on the first drawing sheet "Power diagram". to go back to the first diagram

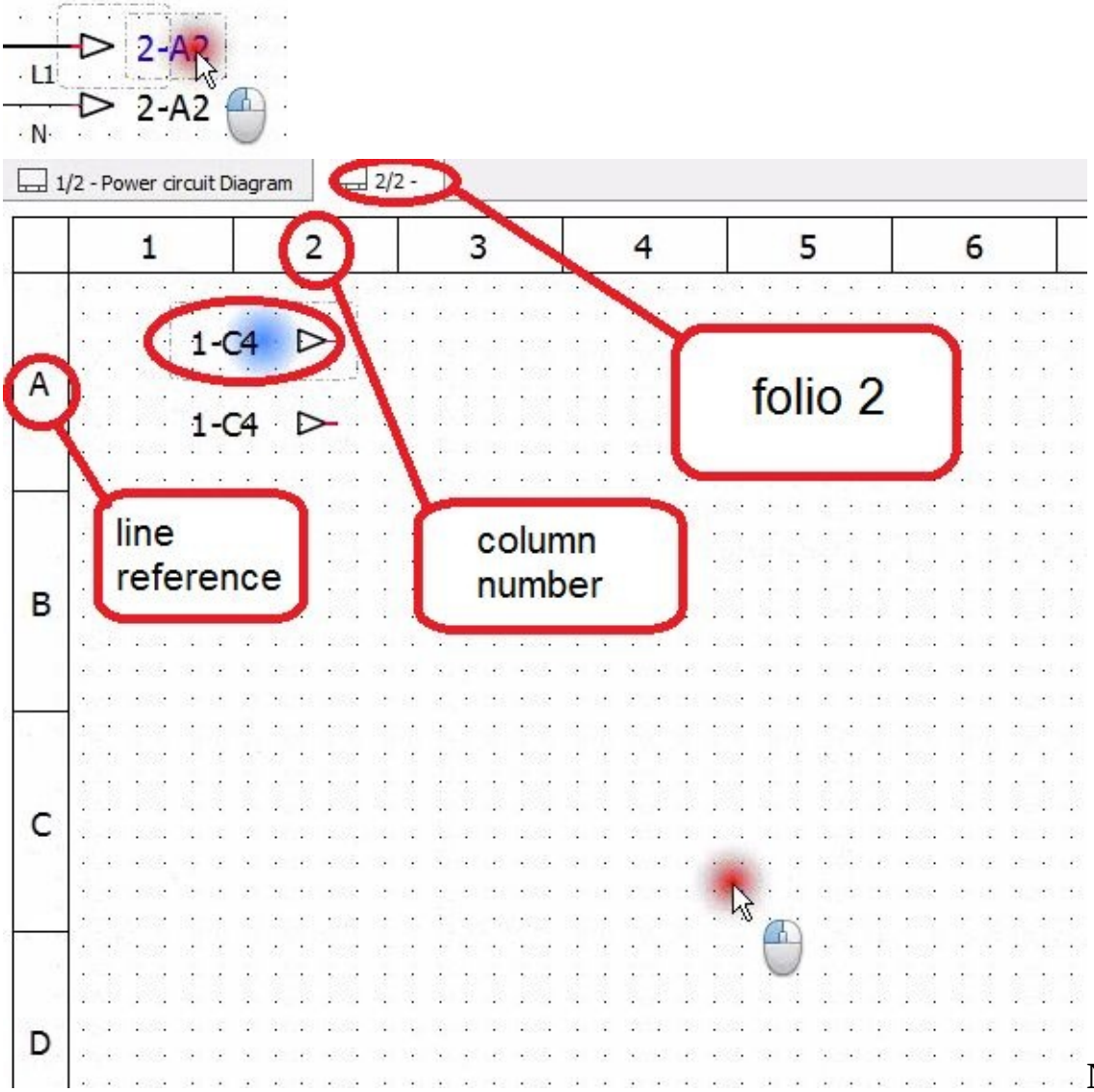
You notice that a text is automatically added next to the arrows.
This text designates the location of the extension of these two.
The first part designate the folio number 2 Folio 2
The second part designate the location A2 Column: 2
Row: A





To understand the utility of cross reference between the arrows double click on the text

Then QElectrotech will show the folio and the location designated by the text.

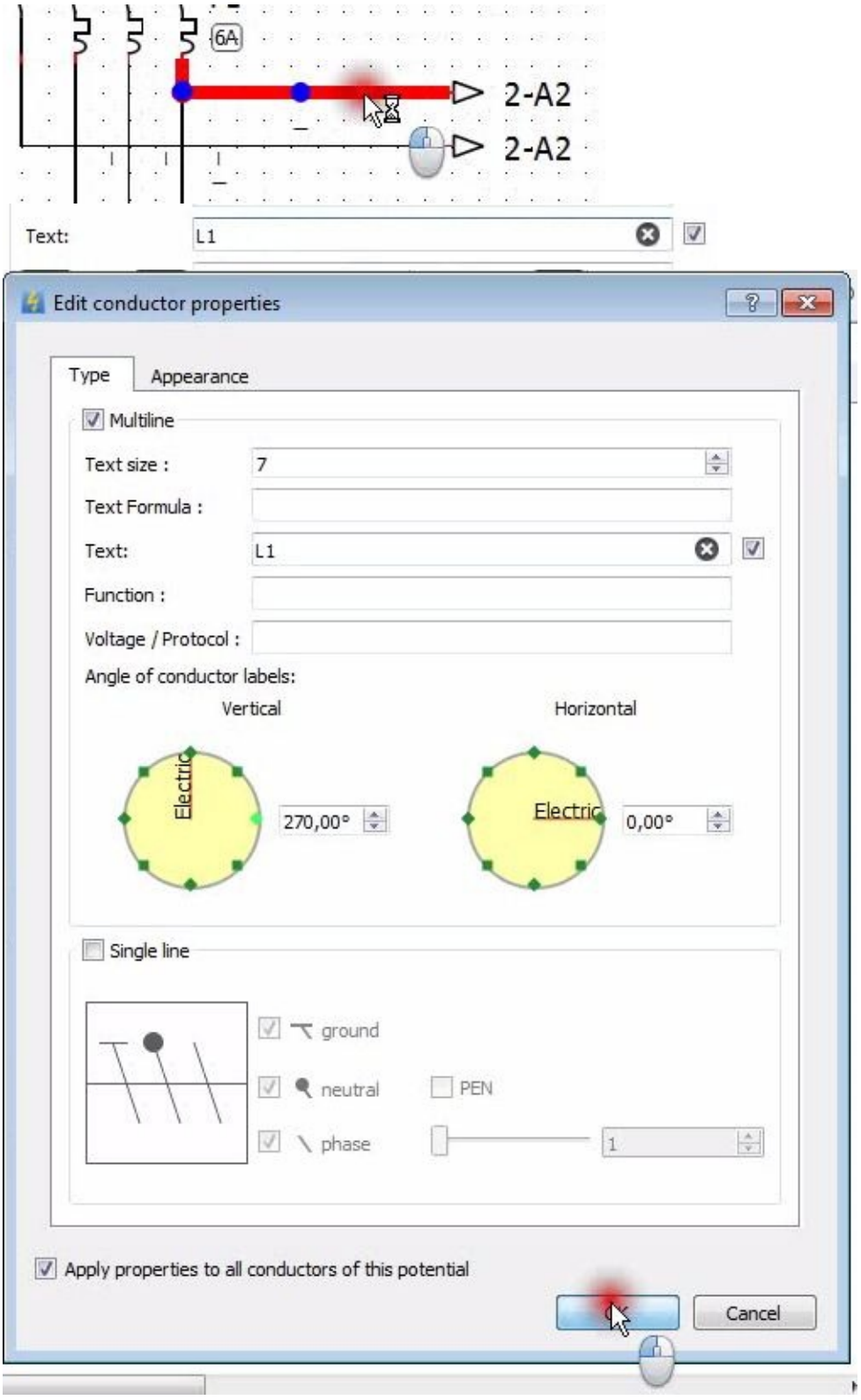


Now we will change the nomination of these two wires

Go to folio1 and double click on the first wire and wait until the dialog "Edit conductor properties" appears

Type in the text box the title of the wire "L1"

Validate the change by clicking OK



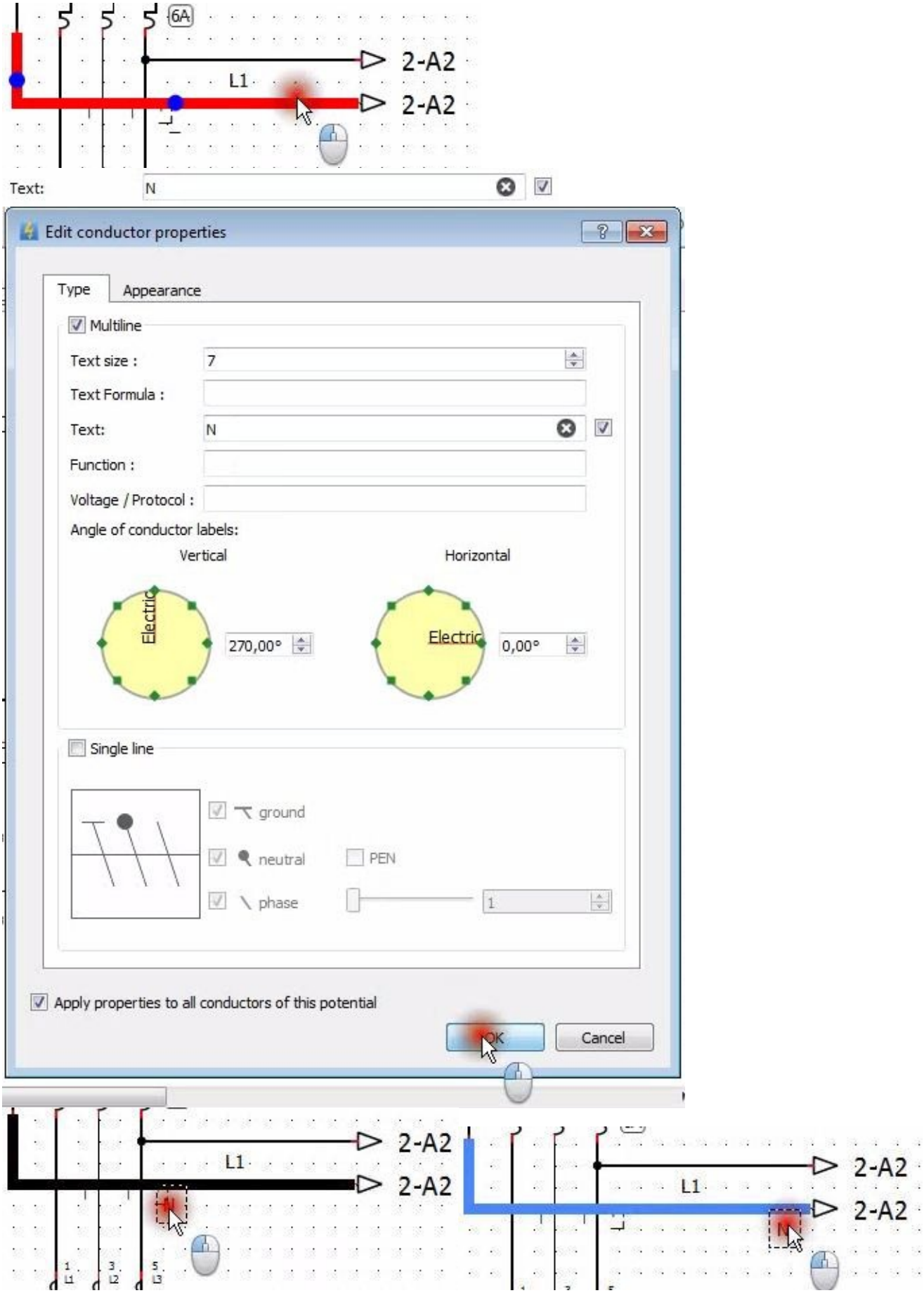
Double click on the second wire and wait until the dialog "Edit conductor

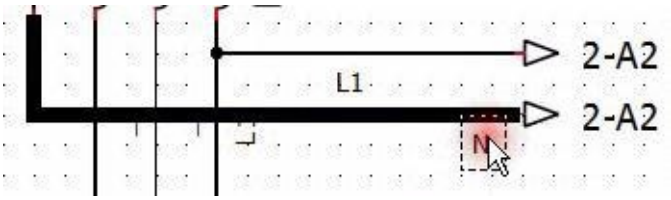
properties" appears
Type in the text box the title of the wire "N"
Validate the change by clicking OK

Move the cursor to select the second wire text designation ("N") Click and hold the left mouse button

Move text

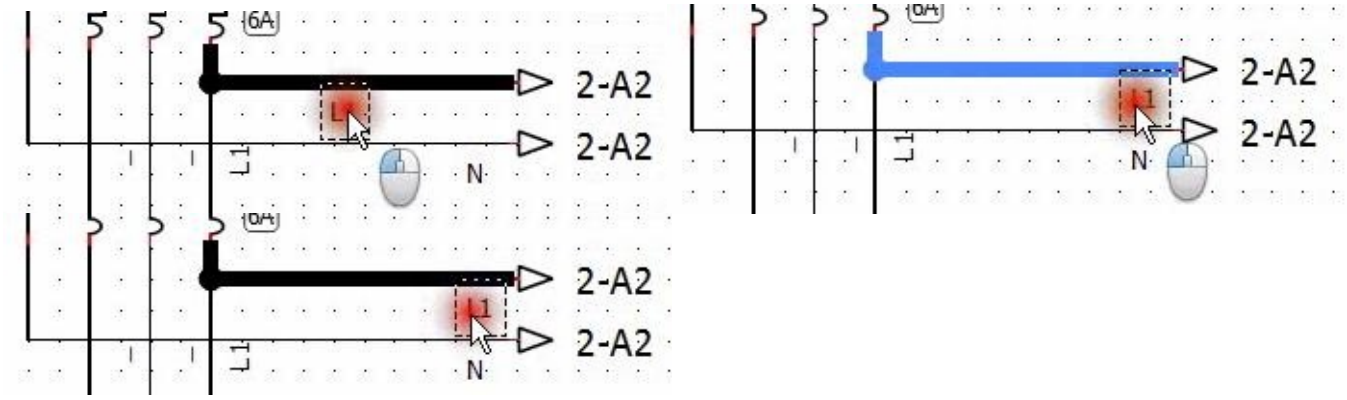
Release the left mouse button





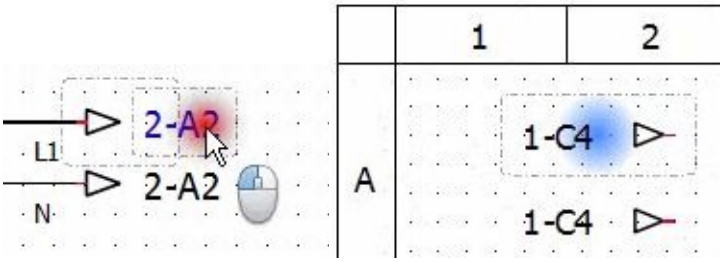
Move the cursor to select the first wire text designation ("L1") Click and hold the left mouse button

Move text
Release the left mouse button



And here we come to make a connection between two drawing sheets of an electric diagram by the elements "Going arrow" and "Coming arrow" this will allow us, subsequently, to navigate easily between the different sheets of the diagram.
The text in front of the arrow indicates the location of the extension in this case we have "2-A2" which means the location "A2" in the sheet 2 (Folio2), knowing that the location "A2" means the cross between line "A" and column "2".
To see more clearly and understand these connections follow the steps below:

Double click on the text in front of the arrow
QElectroTech displays the element linked to the chosen reference



Chapter three III- Finalize the diagram

3 Finalize the diagram:

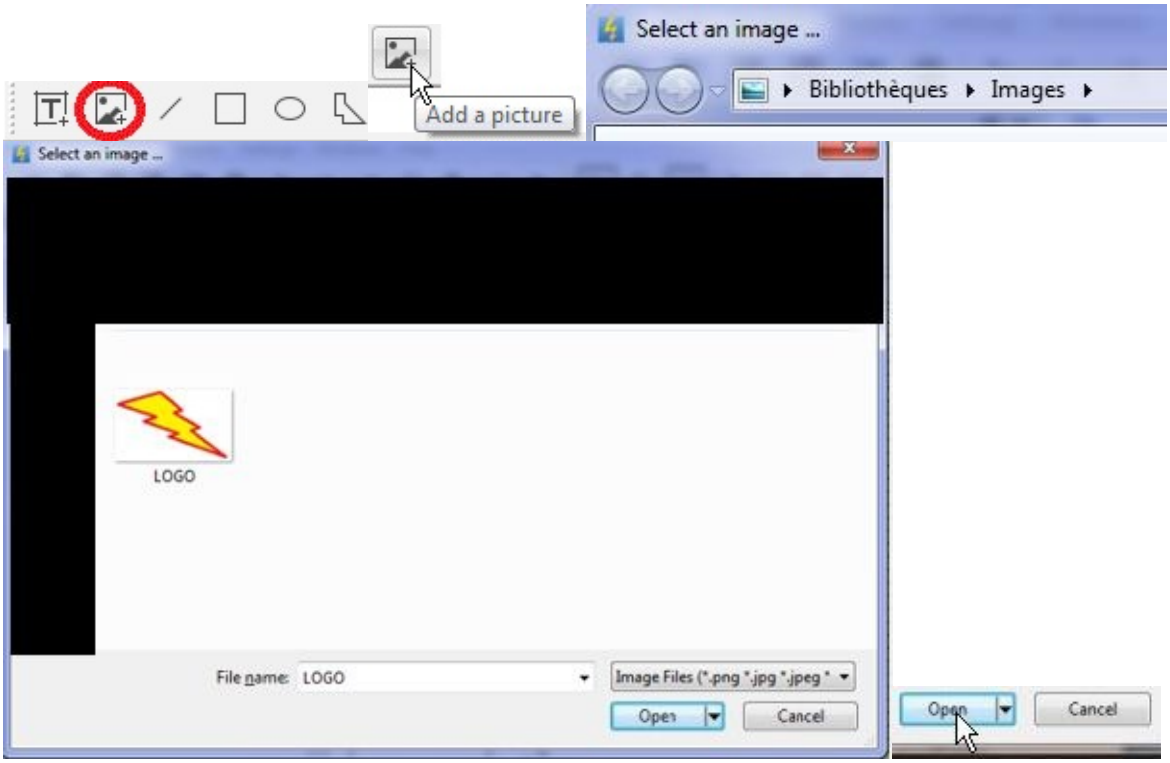
3.1 Insert an image (your LOGO):

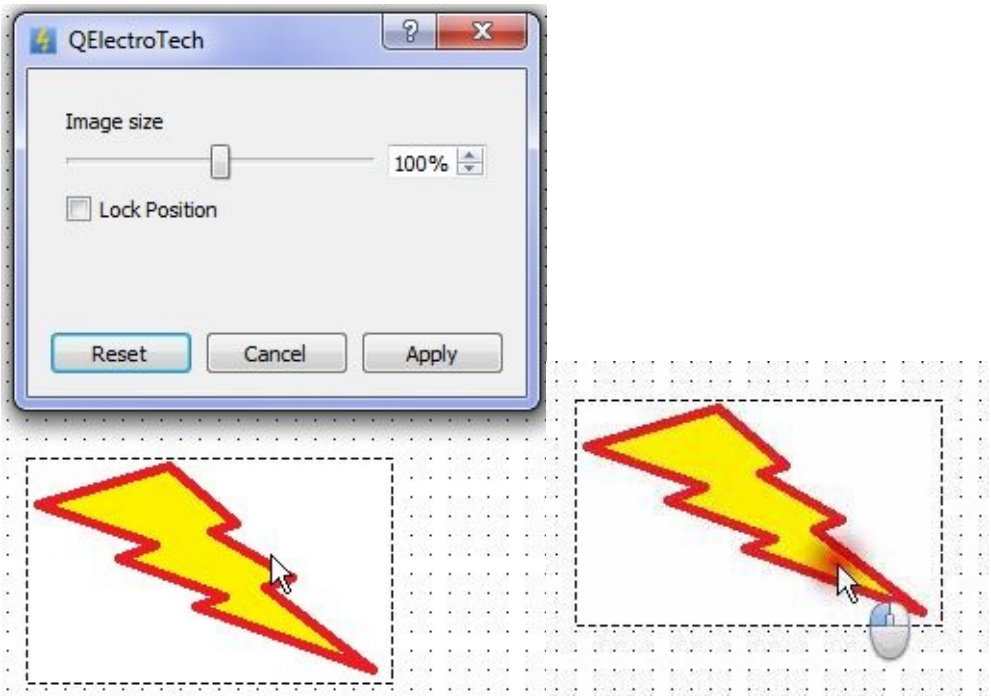
It is possible with QElectroTech to insert images into the sheets of the diagram; thanks to this option you can insert the "Logo" of your company....
Here's how to insert an image:

Click on the "image insertion" icon

Sélectionnez l’emplacement de l’image que vous voulez insérer.
Select image
Click Open
Move the image to the botom of the drawing and clikc

Double click on the inserted image to open the image property box if you need to change its size

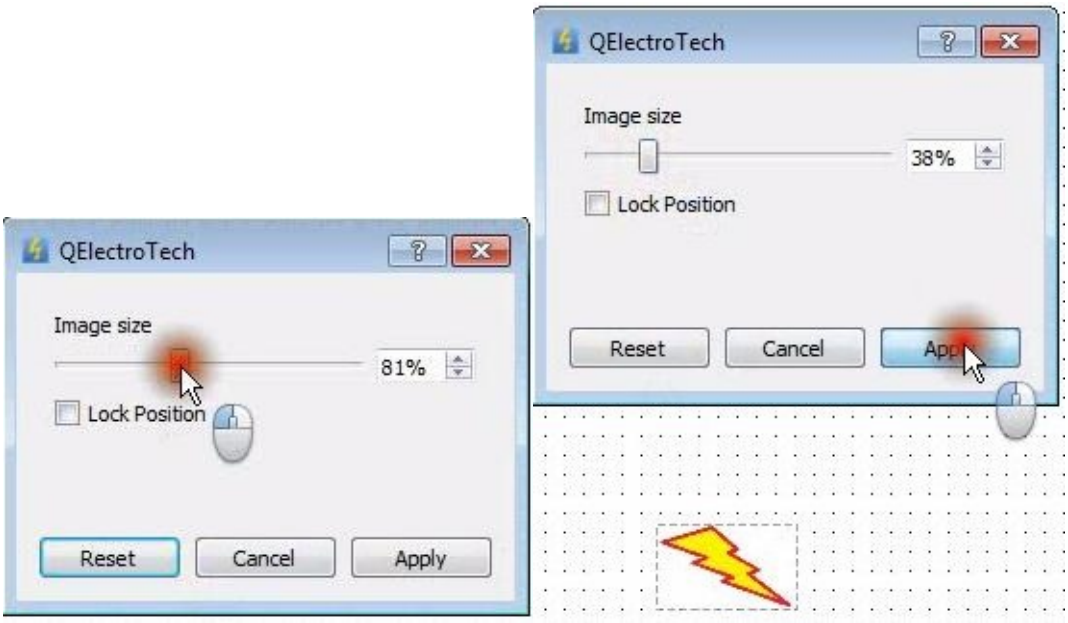


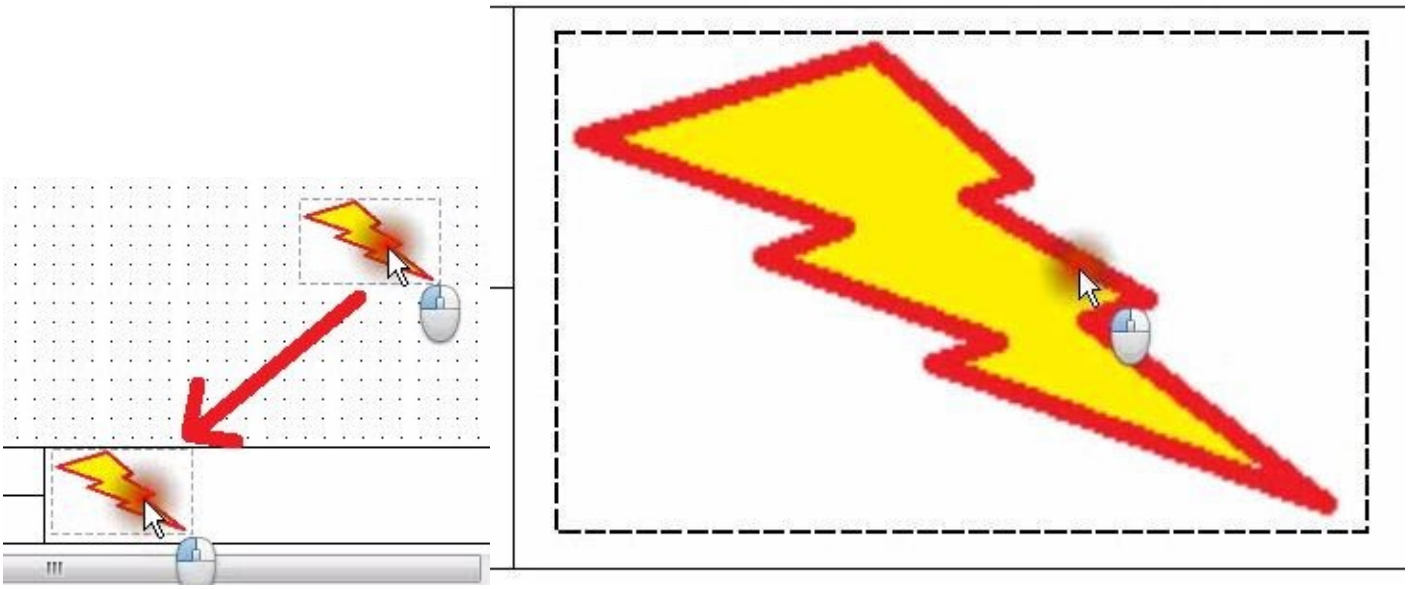


Move the cursor to change the dimensions of the image...

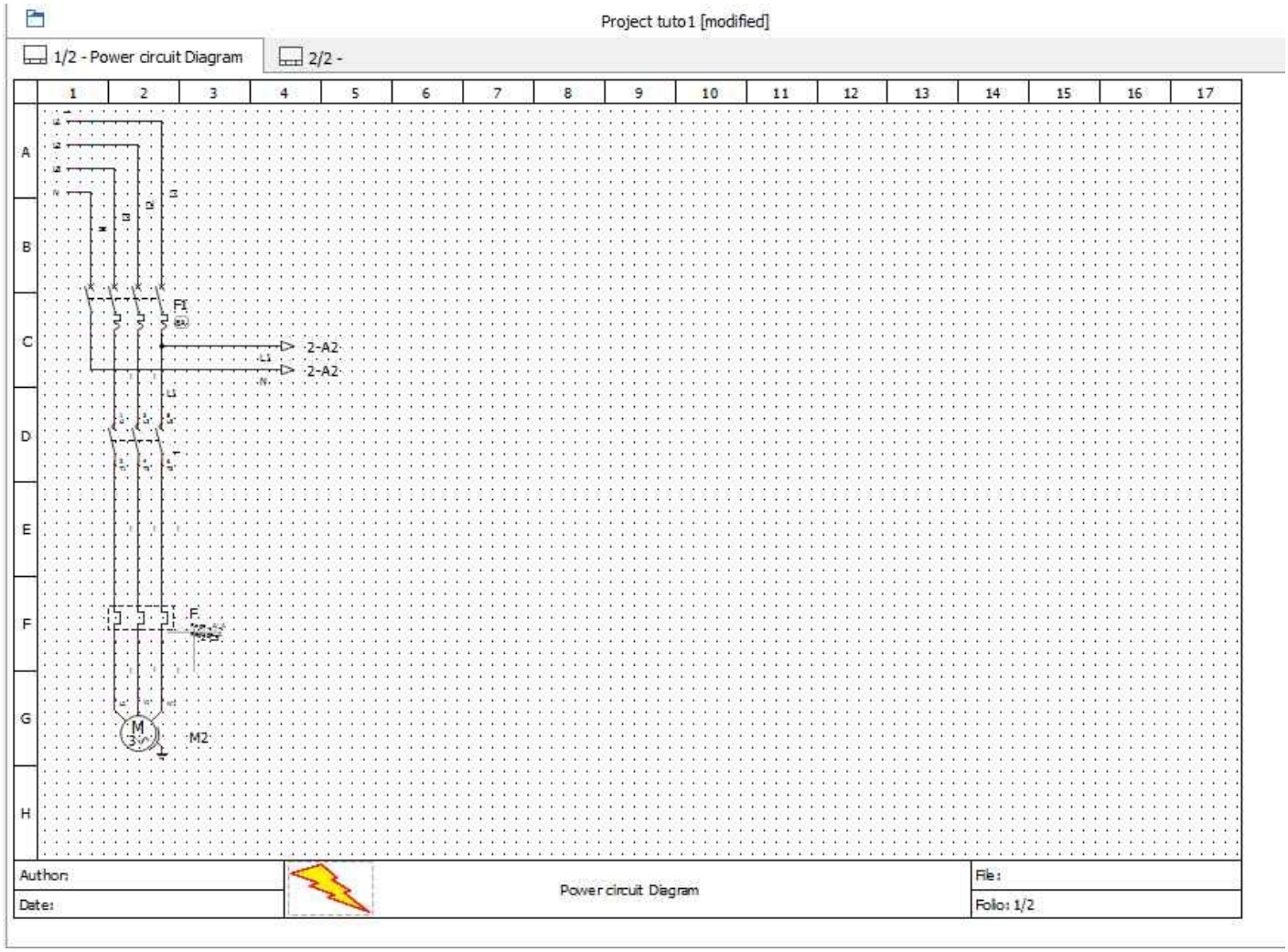
Click Apply to validate the change
Move the image

To see more clearly zoom in and to get more precession click on Ctrl button on the Keyboard while you move the image





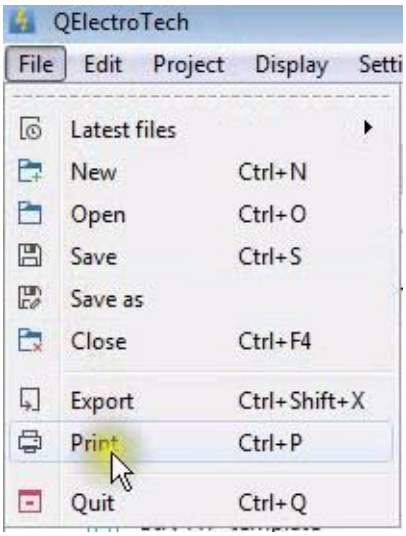
And here is final result:



3.2 Print:

You can print your drawings by following these steps:
Go to the "File" menu and click on
"Print"

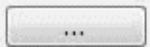
Or use the keyboard shortcut "Ctrl + P"



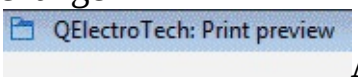
Wait for the dialog box to open You can then choose physical or PDF file printing type

If you choose PDF file printing type You will need to choose the file location.

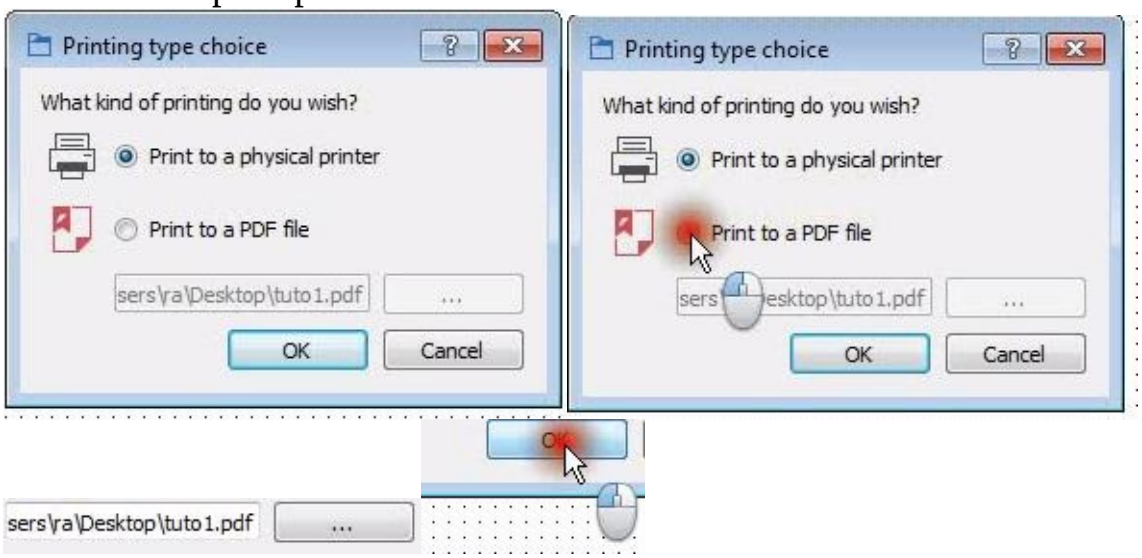
Click on browse

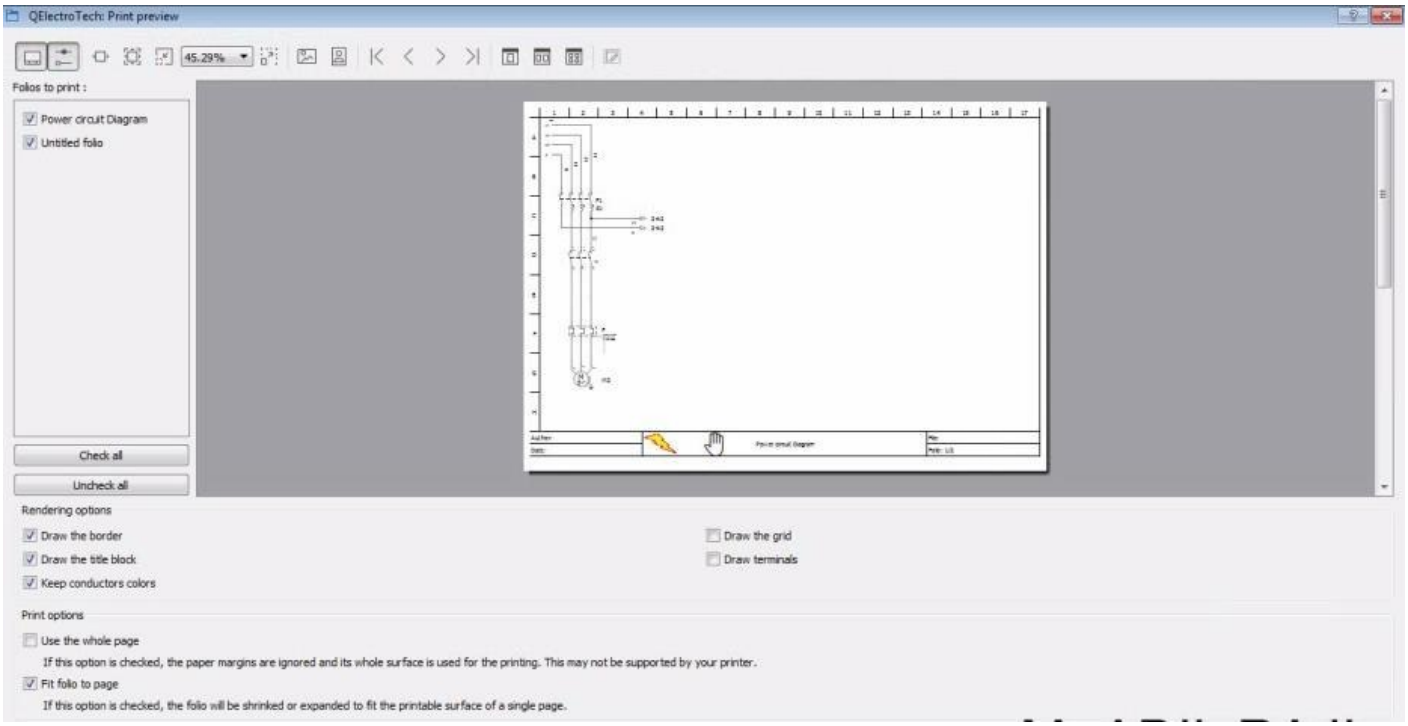


to open the dialog box to select the path to the location of the PDF file
Click “OK” to validate the change

Waite for the print preview 

And click on Print




QElectroTech: Print preview
45.29%
Folios to print :
Power circuit Diagram
Untitled folio
Check all
Uncheck all
Rendering options
Draw the border
Draw the title block
Keep conductors colors
Draw the grid
Draw terminals
Print options
Use the whole page
If this option is checked, the paper margins are ignored and its whole surface is used for the printing. This may not be supported by your printer.
Fit folio to page
If this option is checked, the folio will be shrinked or expanded to fit the printable surface of a single page.

Chapter four IV- Conclusion

4 Conclusion

You can now draw diagrams and easily add images to them.
You can further explore QElectroTech to become familiar with it, and keep in mind that it is really packed with features and is updated frequently.
I will add more tips in the next edition of this guide, thank you for reading this one