

# PLOTXY TUTORIAL

---

## About this tutorial

This tutorial shows the basic operations to obtain the first plots from PlotXY. It shows also nearly everything you will need to make also more complex operations with the program.

Please consider that the maximum efforts have been done to make the program as intuitive as possible, therefore the learning curve should be very steep. I imagine that the duration of this tutorial for you will be around 30 minutes.

The figures whose refer to the Windows' appearance as they are under Microsoft Windows operating system. Under macIntosh they are slightly different but the elements (buttons, tables, etc.) are exactly the same. Therefore any Mac reader will be as comfortable as windows users in following it.

This tutorial is intended for sequential reading. However, in case you strongly want to, you can jump to its different sections by clicking on the links in the Table of Contents, in the following page.

## Contents

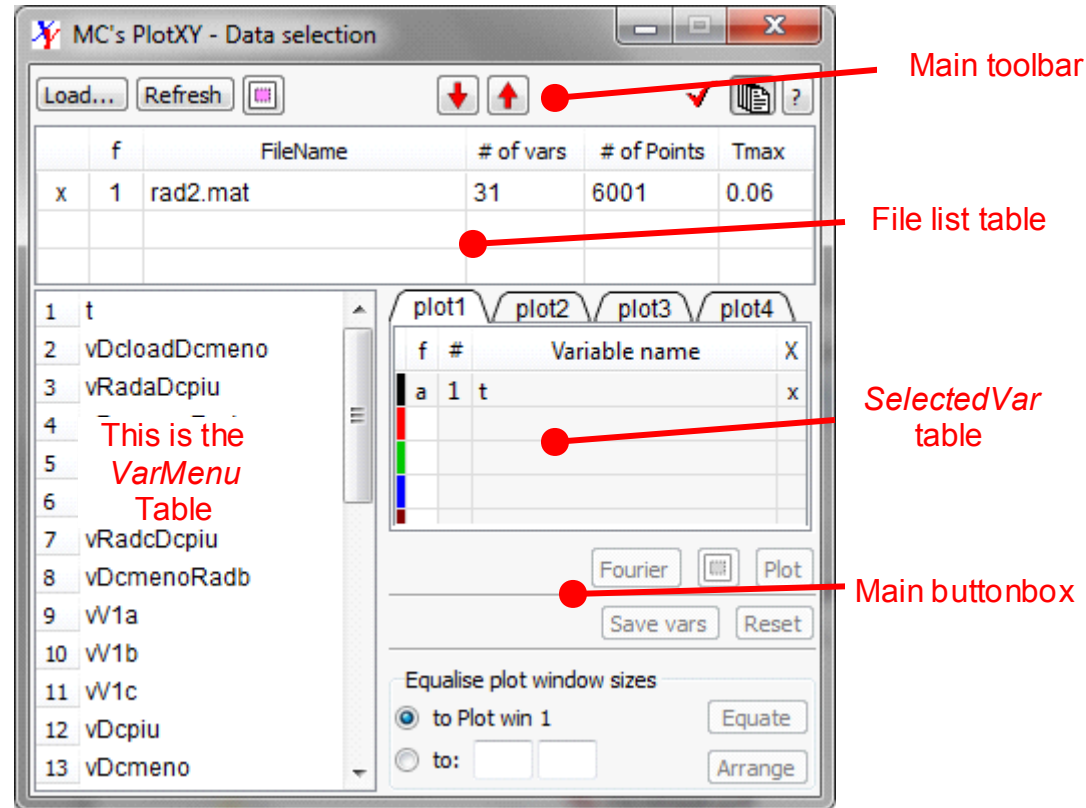
|   |    |
|---|----|
| About this tutorial .....   | 1  |
| Getting to know the main windows and giving names to buttons..... | 3  |
| The DataSelection Window .....                                    | 3  |
| The plot window.....  | 5  |
| The Fourier chart window .....                                    | 5  |
| Doing the first things: a “real-life” experience .....            | 6  |
| Doing some activity using the provided files.....                 | 11 |
| Loading a file and doing first things.....                        | 11 |
| Making , evaluating and managing line plots .....                 | 13 |
| Looking at data values .....                                      | 15 |
| Zooming and keeping zoomed.....                                   | 17 |
| Special plots: with twin vertical axes or X-Y plots.....          | 20 |
| Changing the plot type and appearance .....                       | 22 |
| Exporting selected variables into a new data file .....           | 28 |
| Exporting plot into system clipboard or SVG/PNG.....              | 28 |
| The “function of variables” feature.....                          | 29 |
| Creating a Fourier Plot.....                                      | 33 |
| Using several program windows .....                               | 34 |
| Saving Program State.....   | 36 |
| Printing on paper or pdf.....                                     | 36 |
| Automatic loading of files.....                                   | 37 |
| Additional information .....                                      | 37 |
| Customizing the program behaviour .....                           | 37 |
| Automatic Units of measure and prefix.....                        | 38 |
| More on loading files .....                                       | 39 |
| What else?.....   | 39 |

## Getting to know the main windows and giving names to buttons

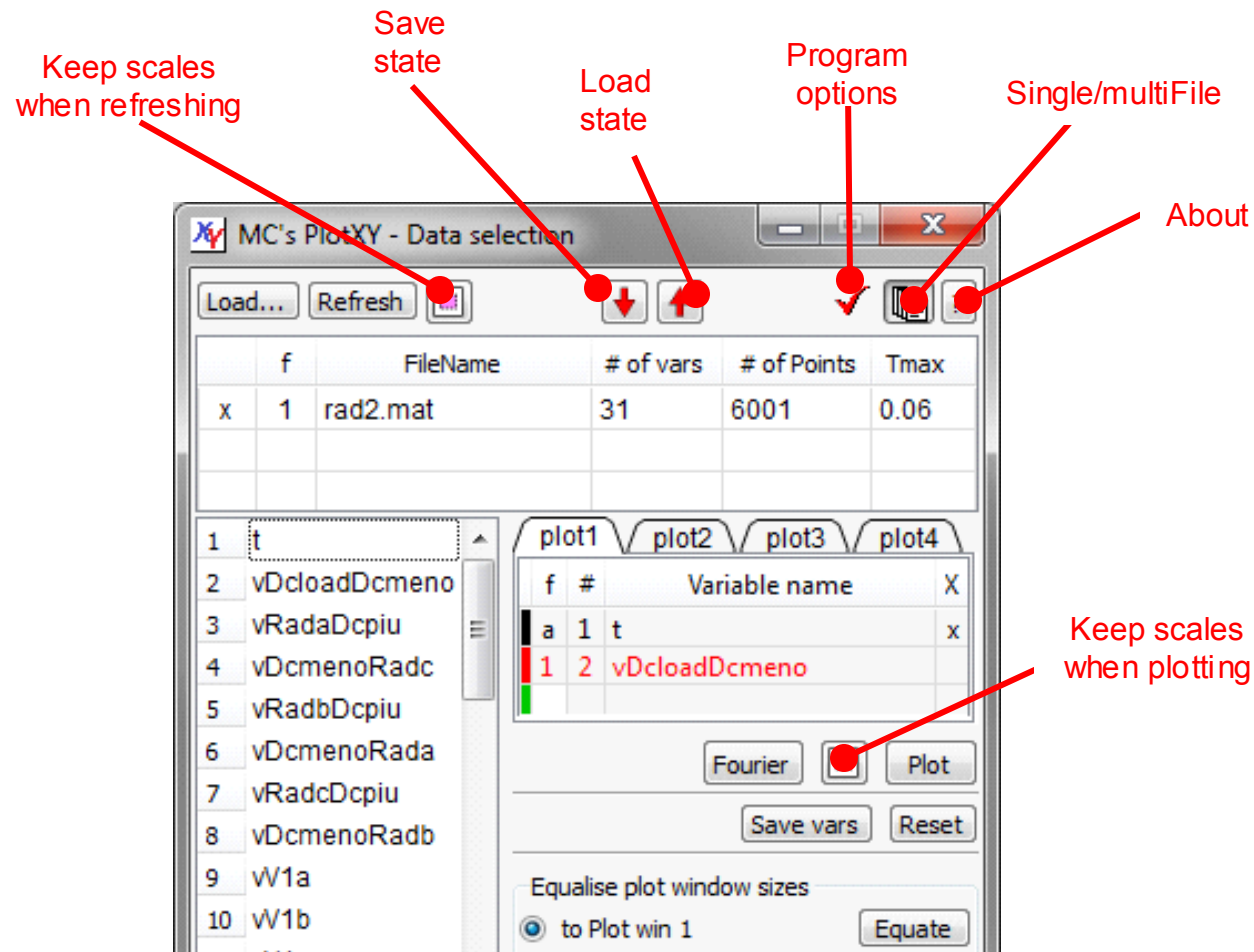
Before starting doing things, better is to become acquainted with the three program main windows, and to give names to their buttons.

### The DataSelection Window

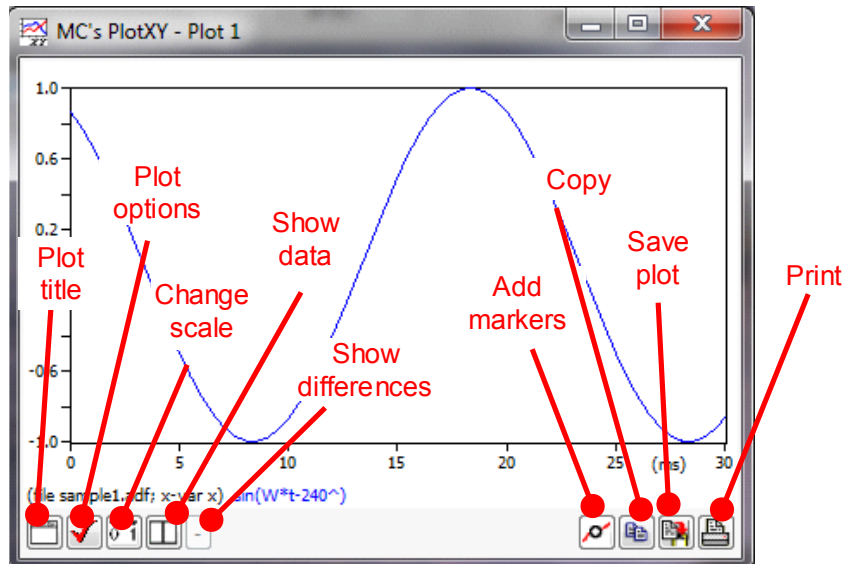
Once PlotXY is started, the first window to appear is the one shown below, that is the main program window.



Below you can find specific names of the toolbar tools



## The plot window

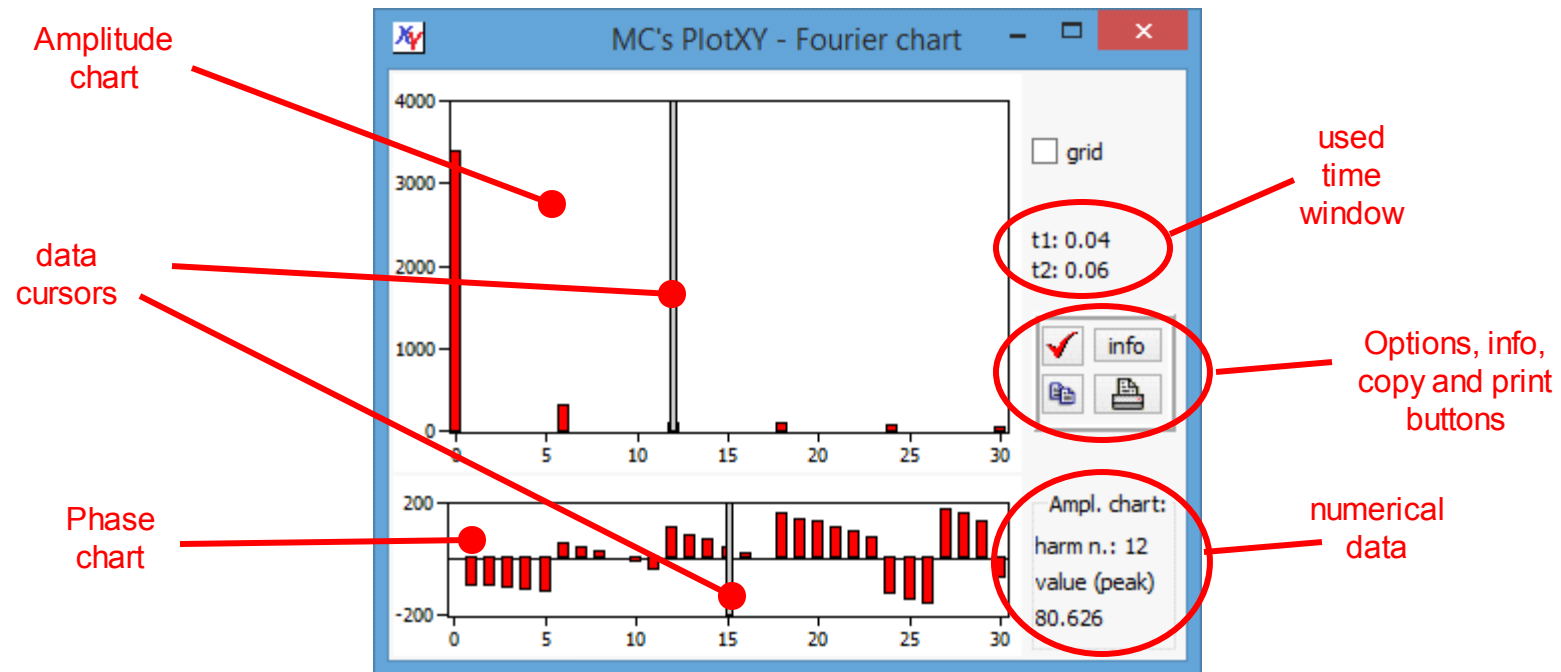


When, in the Data Selection window the plot button is clicked, the plot window is displayed.

It has the appearance shown in the picture aside (in which a simple plot from the supplied file "sample1.adf" is drawn), where names of the toolbar buttons are also given.

## The Fourier chart window

The program is also able to make Fourier analysis (more precisely the i.e. Discrete Fourier Transform - DFT) of periodic signals and to display the corresponding amplitude and phase spectra. This is done using the Fourier Chart window, that is shown below along with the name of its main elements.



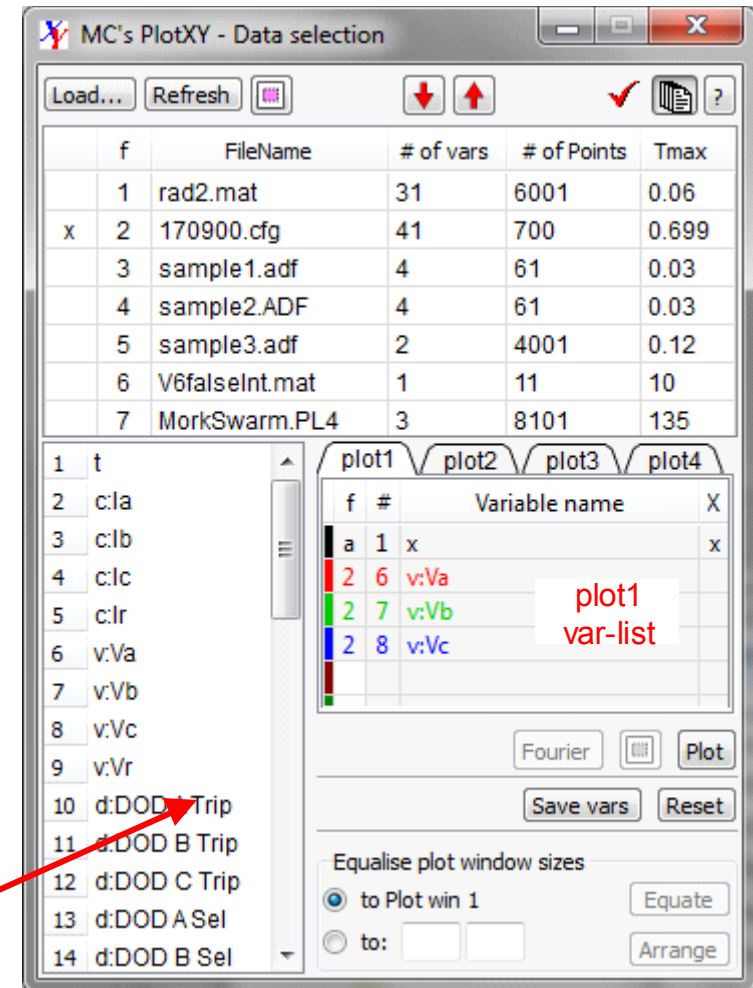
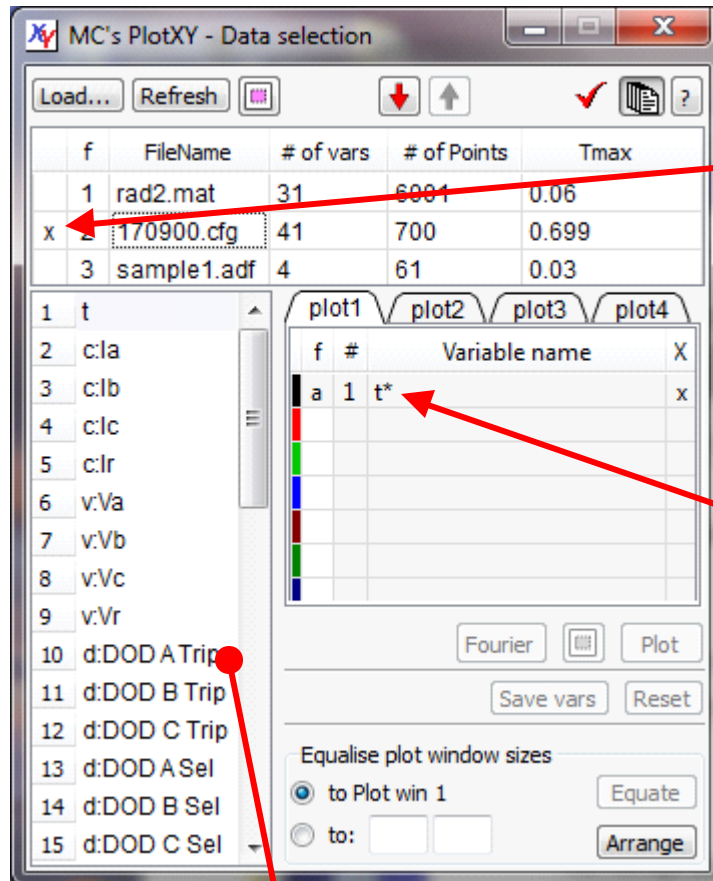
## Doing the first things: a “real-life” experience

The first thing to do is to load one or more files. Here I show how it works using my own files, just to give ha hint about a real-life experience with the program. In [Working together using the provided files](#), instead I will propose actions that you can repeat using the sample files provided.

There are two ways of doing this:

1. Using the Load... button
2. Using the drag&drop feature of the operating system: looking at the file name in their own directory, selecting one or more of them, and dragging them onto the main program window.

Once some files have been loaded the main program window can for instance have the appearance shown below at the left side of the figure (three files) or at the right side (seven different files).

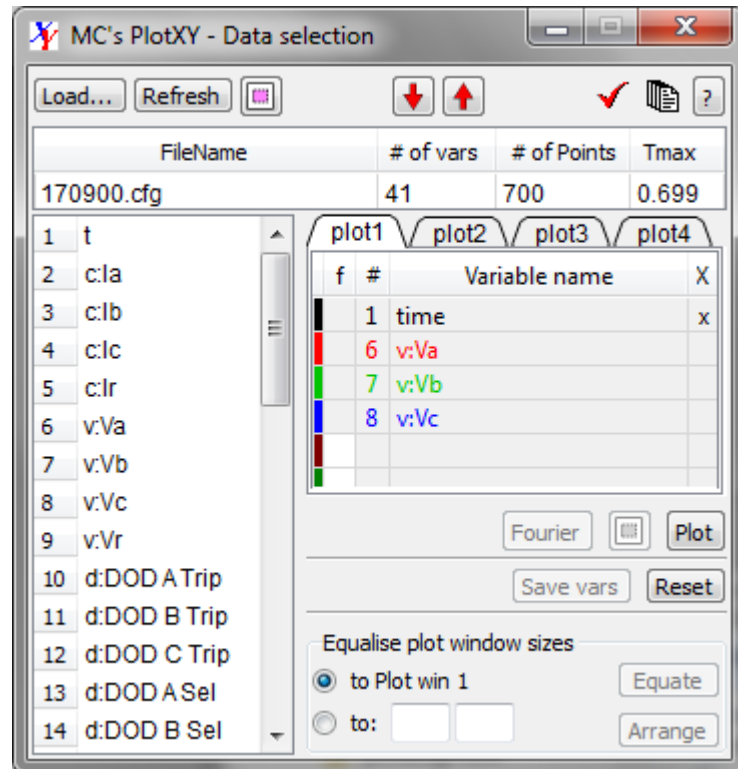


Note that the program automatically selects the first variable as the “x variable” (the one that should by default correspond to the horizontal axis). If all the loaded files share the same name on the horizontal axis, this one will be chosen.

If all the first names begin with the same first character a name containing that character and “\*” will be used. For instance, if all the variables begin with ‘t’ we will have a “common time” variable indicated as “t\*” (see figure left above). Note that based on the first character the horizontal variable will be attributed an unit of measure, according with the following convention:

- Names beginning with 't' will be treated as being time (unit of measure is second)
- Names beginning with 'f' will be treated as being frequency (unit of measure is hertz).

If the first names in the selected files var-lists do not have all the same first character, the program gives to this common variable the name "x" and gives up trying to understand which unit of measure it might have,



There are times, however, in which the user wants just to deal with a single file. In this case he can switch to the single-file mode, simply by clicking on the second rightmost button on the main toolbar. This way he will save valuable screen space, and have some additional features that are available only in single-file mode.

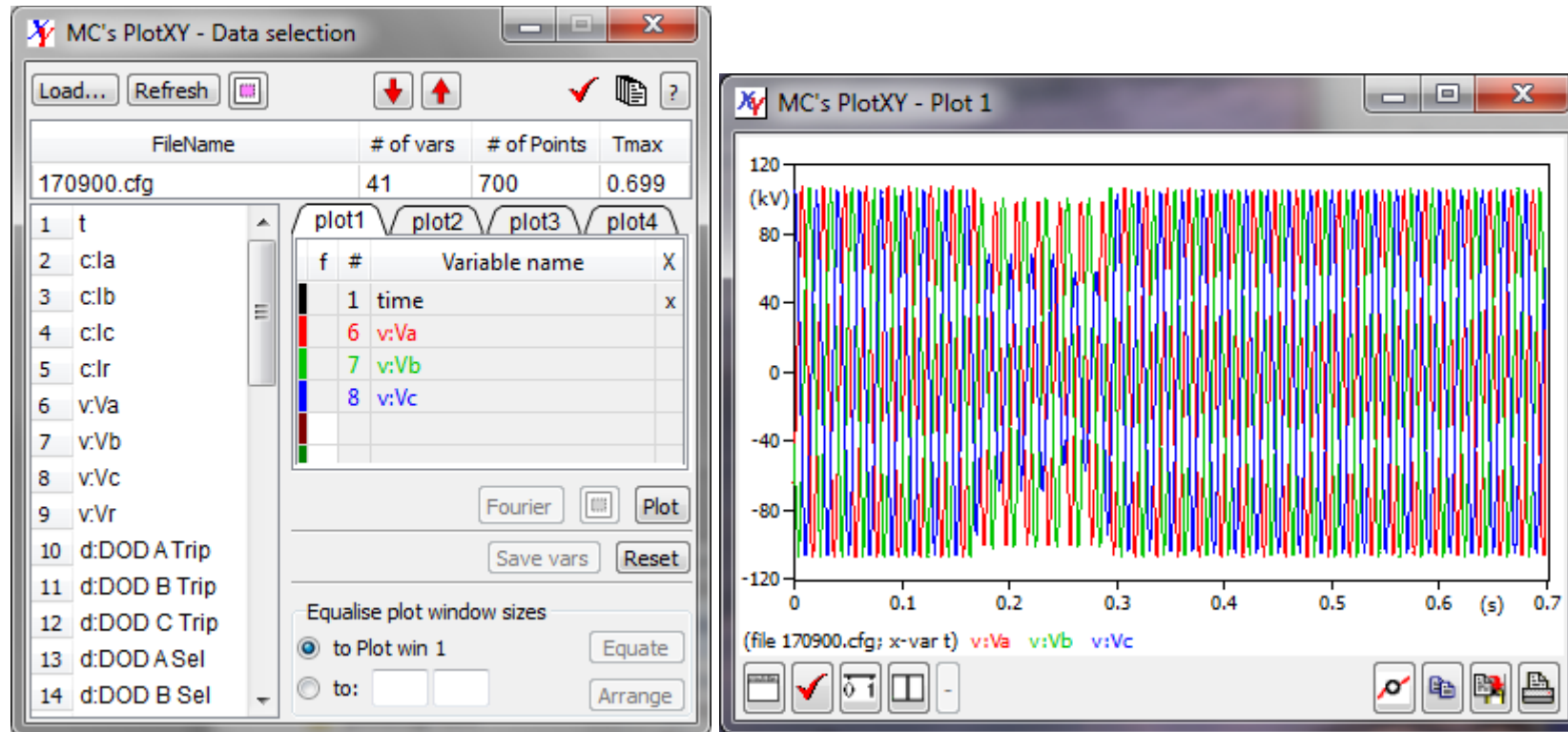
Switching into *singleFile* mode does not unload any of the already loaded files. When I clicked on the second right-most button of the toolbar from the situation shown in the right window just above here, I got the appearance shown aside (I have also reduced the window's height by acting as usual on resizable windows).

Now it is time to select and plot variables.

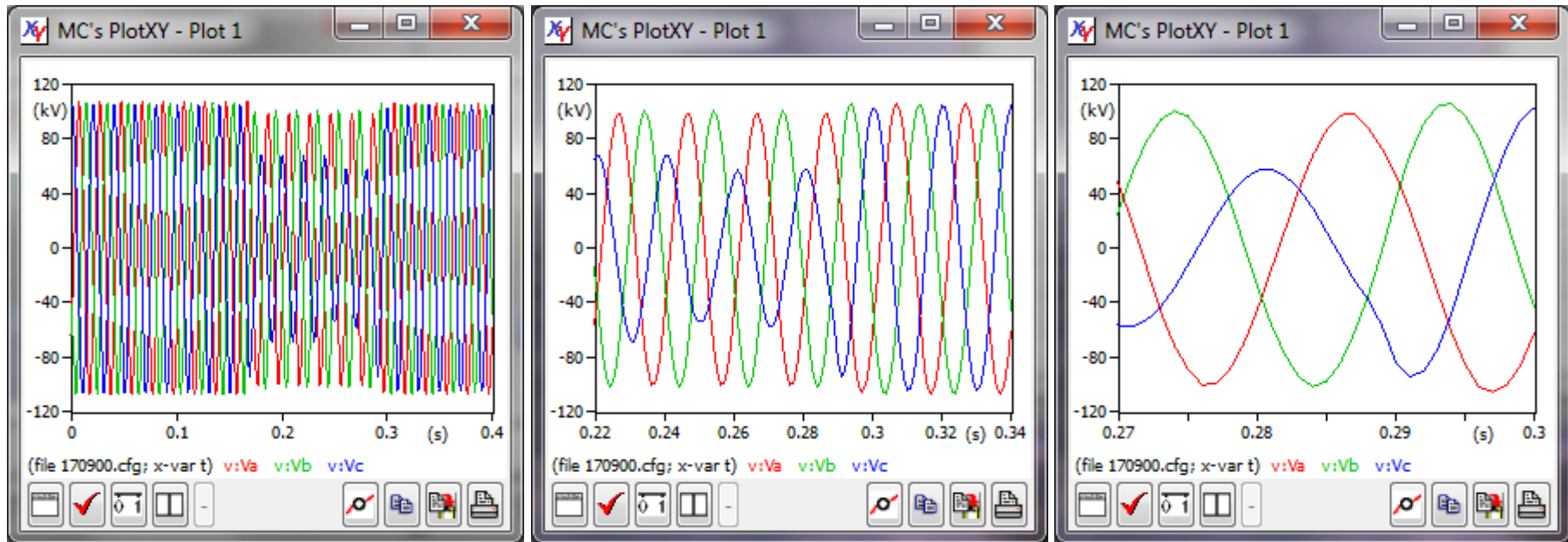
From the figure aside we now select (by clicking onto the respective names) variables #2, #3, #4. The window will now appear as shown below at left.

Once the variables have been selected, they can be plot simply clicking on the "Plot" button. In my example the window shown below at right appears.





As often, in this case the plot is too cluttered to see details. However, it is very simple in PlotXY to zoom: simply click and drag. To zoom out, right-click and choose whether zoom just one step or totally. Below three progressive zoom levels of the same figure are shown in three windows that, for compactness were been previously horizontally shrunk.



You might have noted a few characteristics of the plots that are produced:

- 1) Minimum and maximum numerical values on the axes are always “round” numbers. Indeed a rather sophisticated algorithm is present inside do avoid nasty numbers on the axis such as 1.2345 or 100000 or 0.000001
- 2) The unit of measure of the quantities is automatically set to kV. This is based on the variable names. If the auto-naming feature is not disabled, all variables whose name begins with “v” is taken to be a voltage. Also, time is acknowledged as such and its unit set to seconds.

The rules used for auto-setting of units of measure are as in the following table:

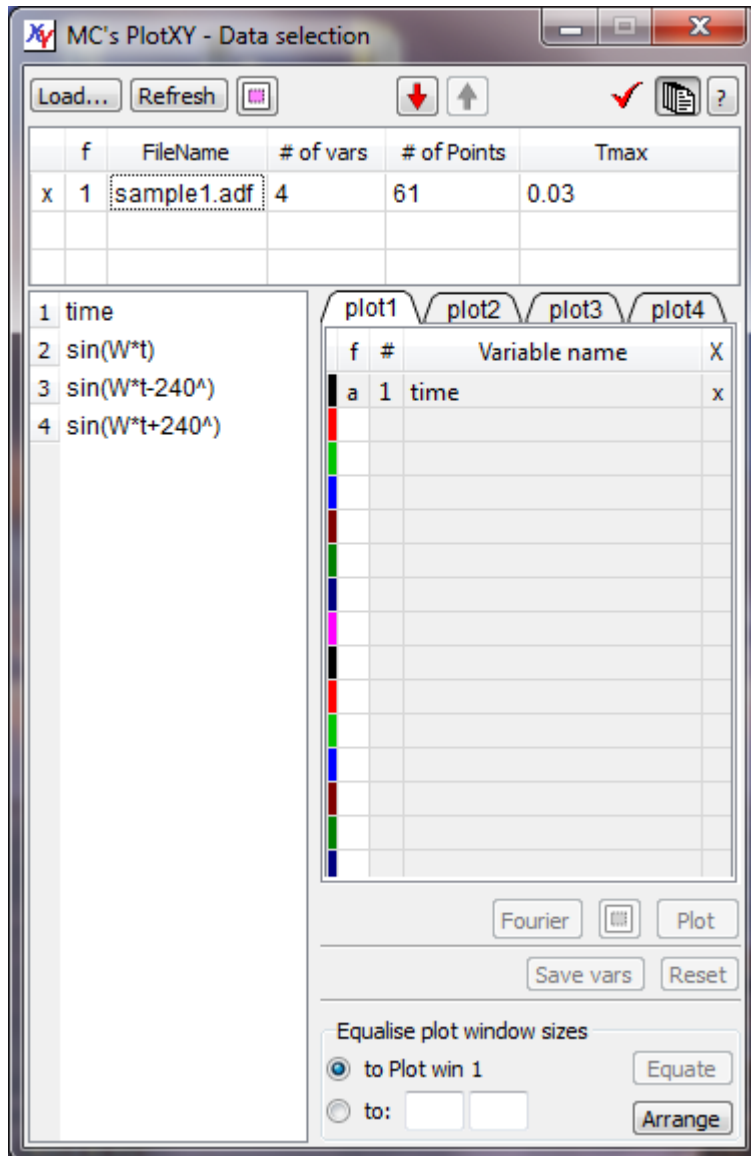
| <i>First character of name</i> | <i>quantity</i> | <i>First character of name</i> | <i>quantity</i> |
|--------------------------------|-----------------|--------------------------------|-----------------|
| “t”                            | time            | “a”                            | angle           |
| “v”                            | voltage         | “p”                            | power           |
| “i” or “c”                     | current         | “e”                            | energy          |

When these units of measures are used, also the standard prefixes are used as well (e.g. u for “micro”, m for “milli” k for “kilo”, M for “Mega”, G for “Giga”, etc.).

Now that you have a first idea of how the programs work and how the produced plots look like in a real-life case, let us make together some plots step-by-step using the enclosed sample files (from sample1.adf to sample3.adf).

## Doing some activity using the provided files

### Loading a file and doing first things



If you have gained access to this tutorial, I can assume you have already installed the program and understood how to launch the program.

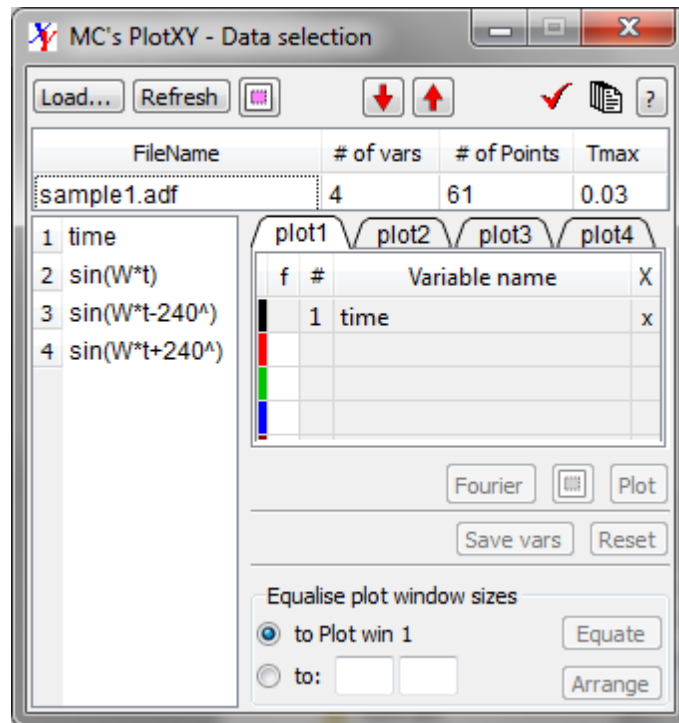
However, to launch the program the best way is to use a link you've created linking to PlotXY.exe (that you've put in a directory of your choice) and click on it.

When the file is loaded you have two options to load a file (either using the button (Load...) or dragging a file onto the PlotXY window from the directory where it is.

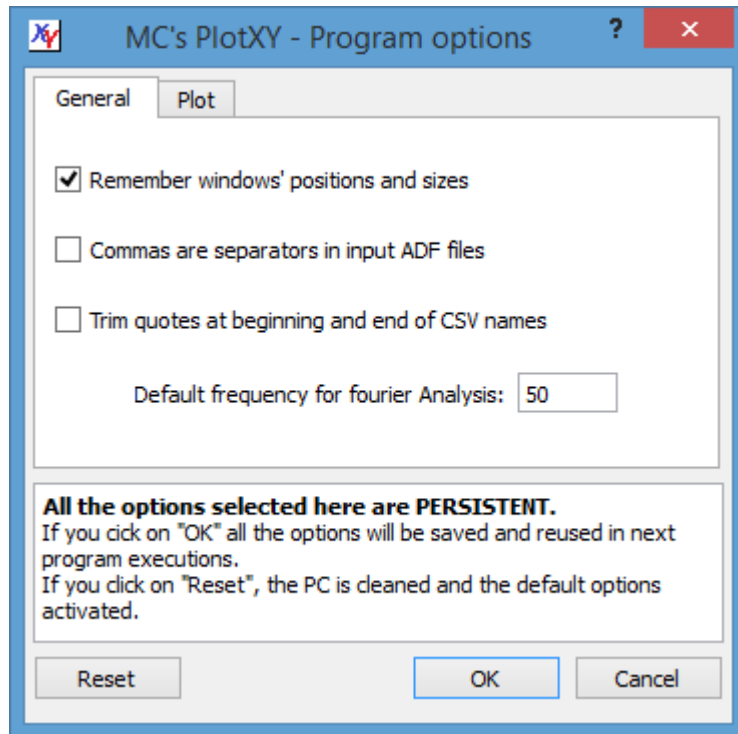
Once you've loaded the program and loaded sample1.adf the program window looks like the one shown aside.

Since the file is compact and just one, we can reduce the window size to reduce its space occupation on the screen. To reduce the window size se use the usual way for resizing windows in computers using the window's borders.

Moreover, we choose to display in the file table only one file we click on the Single/multifile button in the main toolbar



The corresponding windows will now look like as shown aside: the window is much smaller and saves much of your precious desktop area.



Naturally, if we commonly use small numbers of files and variable and plots, we want to use small program windows as default, without having to resize it anytime we load the program. To do this, we can set the option “remember windows’ positions and sizes”. The program options are accessed clicking on the third button from the right in the main toolbar showing a check-mark as symbol.

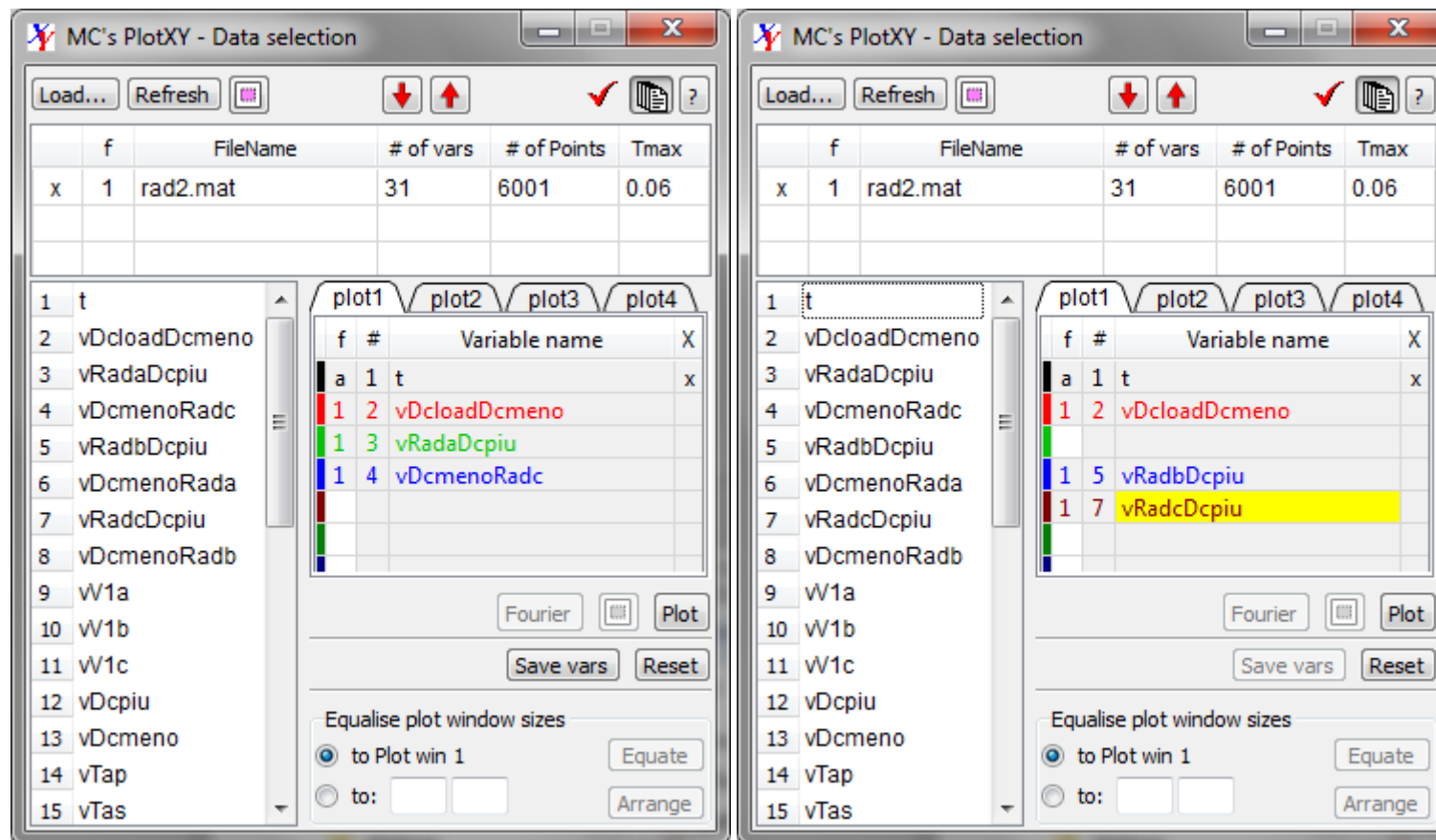
Several program options can be set. Here I did show just the one interests us now. Note that the program settings are written as system settings in a region of the computer storage space the operating system chooses. Under Microsoft Windows it is the Windows’ registry.

Although I don’t advise this ;- ) you might want to remove the program from your hard disk and leave the computer completely clean, To do so, you have to open the program options window and click on the “Reset” button.

### Making , evaluating and managing line plots

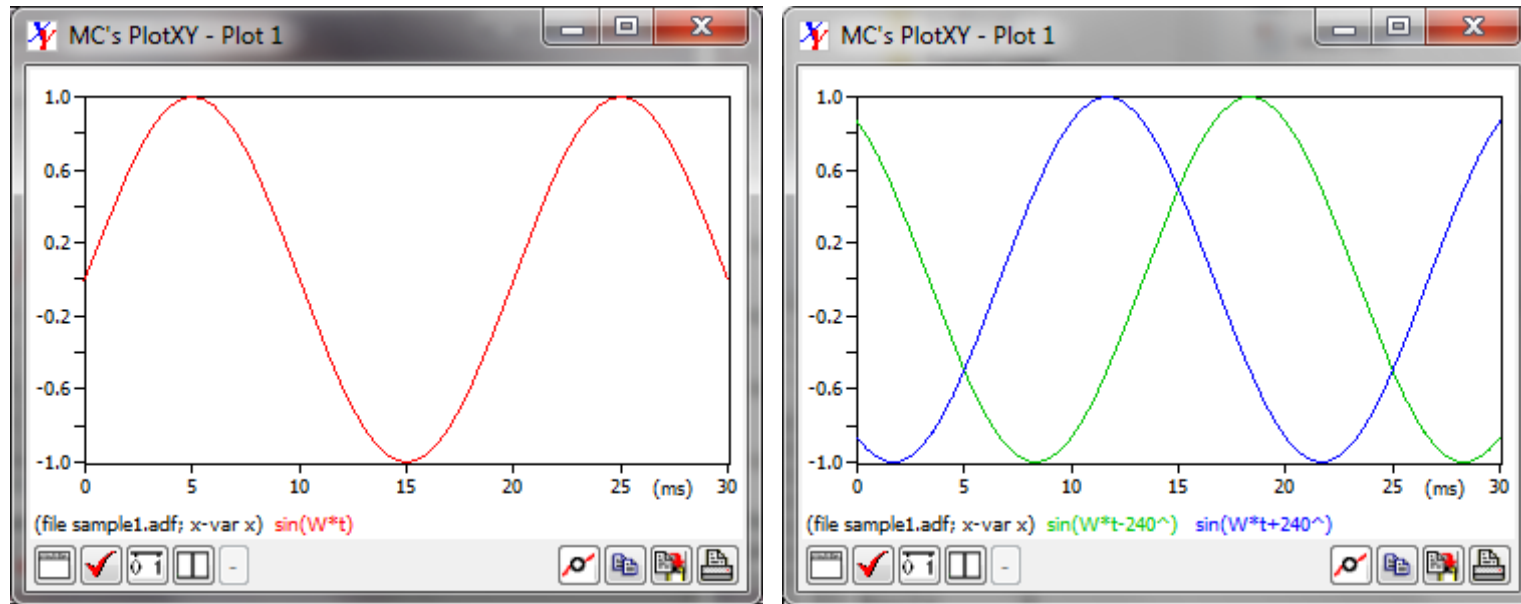
Now we want to select a variable and show its plot. To select a variable you can click on its name in the *VarMenu* table, or drag &drop it in the *SelectedVar* table. In the first case the variable will get the first available colour, in the latter you can choose the variable colour by choosing the appropriate row.

In the figure left below a few variables from the supplied file rad2.mat re elected by clicking on their names; in the one at the right side their names are dragged and dropped, so that the wanted colours are used (the last dropped has a yellow background). The plots will have the same colours as the shown variable names.



You can also select several adjacent variables at a time: a *group* of variables. You have to keep the shift key down while clicking on the first (the uppermost) variable of the group, and then click normally on the last one.

Once one or more variable(s) is (are) selected, click on the “Plot” button to see the plot. Two examples from file sample1.pdf are below: please reproduce them as a very simple exercise.



We see that the plot contains also a legend (shown below) containing the variable names.

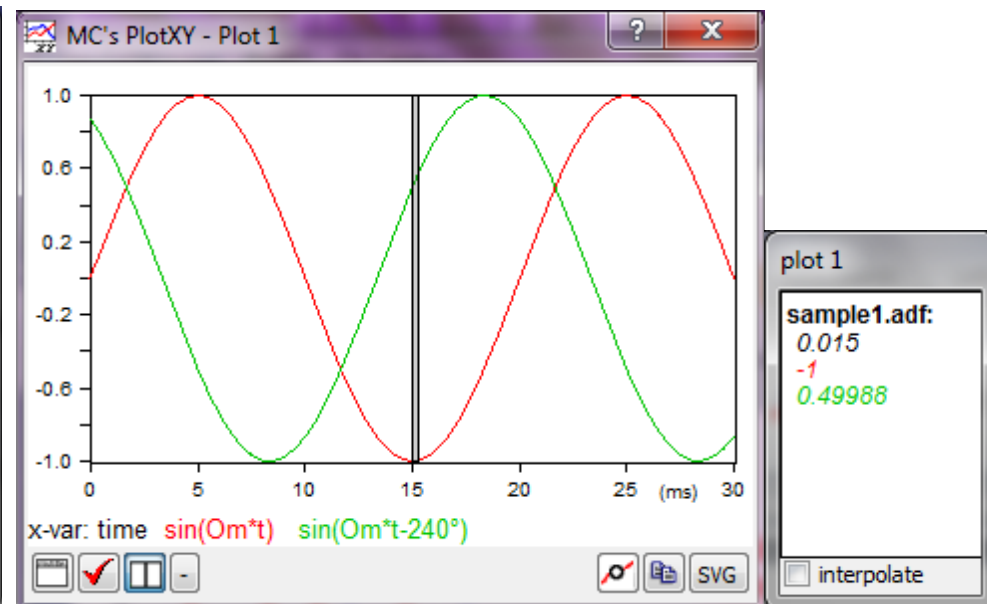
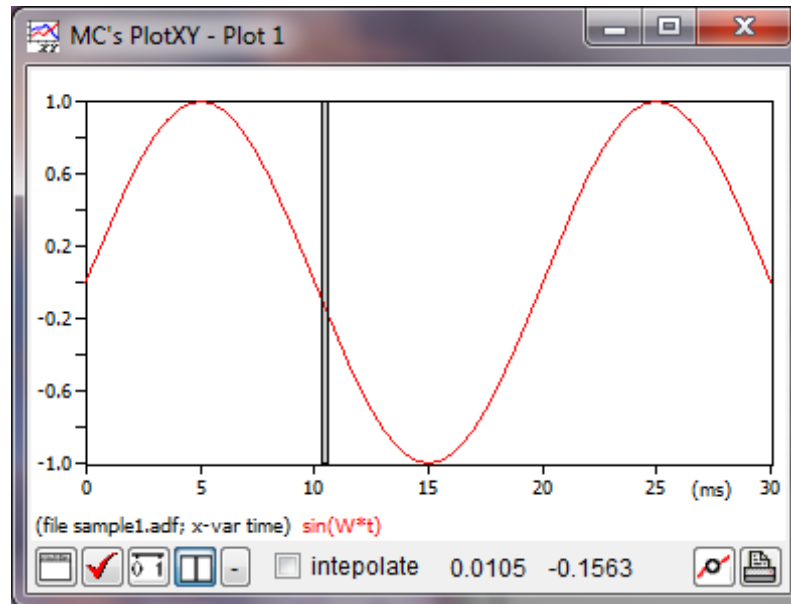
You can zoom the produced plots at will, as we've learned in the [Doing the first things: feel a real-life experience](#).

Now we perform additional actions.

### Looking at data values

Consider the plots in the right picture here above. We now want to see some numerical values.

To do this, click on the Show *data* button. What happens in the cases of a single curve or multiple curves is different. Try! You will get something like what you can see down here.

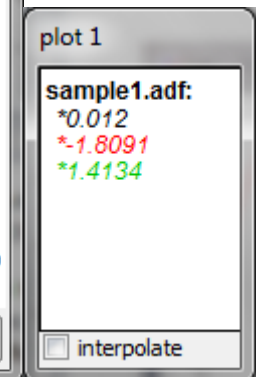
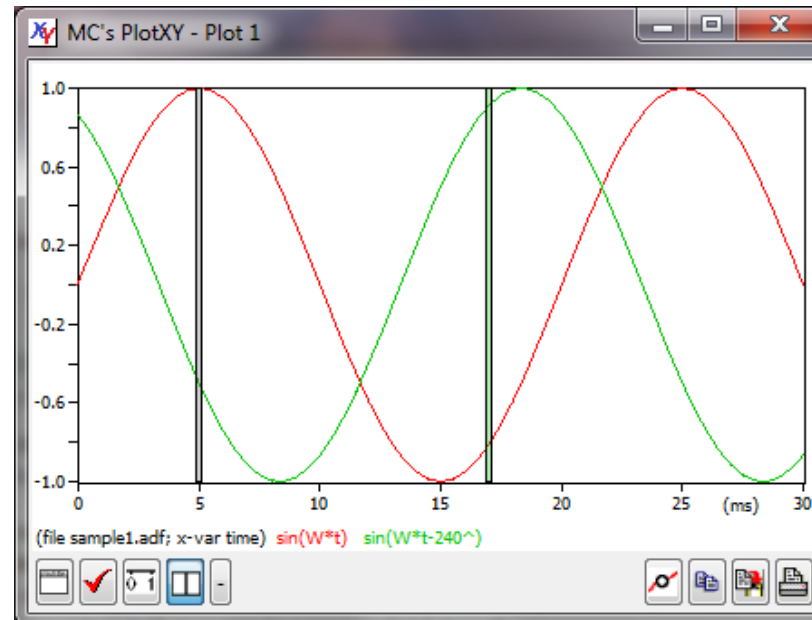
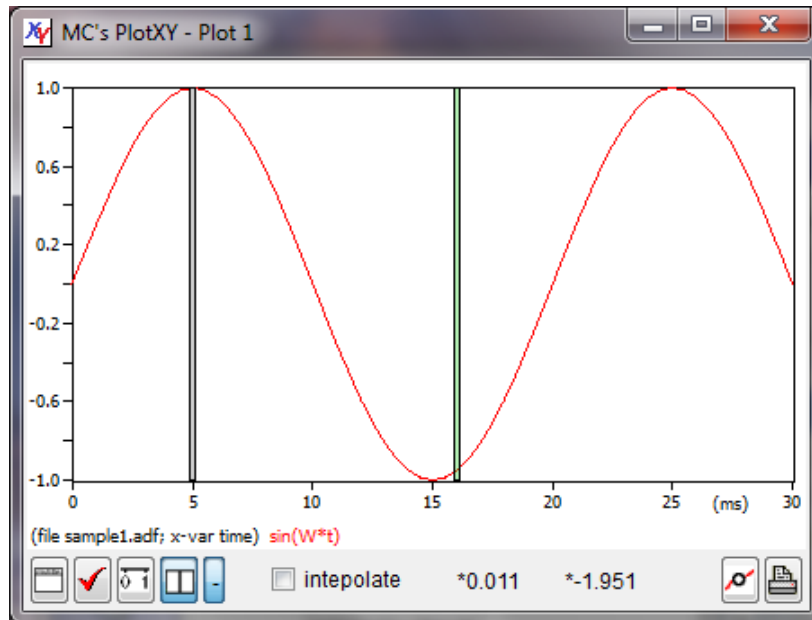


In the left window the values are directly shown in the bottom toolbar (time first), while in the right one, because more space is needed, an additional small window (the *data browse window*) is created to show the numerical values.

In both cases normally the numbers shown are exactly those present in the inpt file; however, if the “interpolate” box is checked, linear interpolation is made between adjacent points, check sample1.adf file: you will see that corresponding to  $t=0.0105s$  the  $\sin(Om*t)$  value is -0.1563.

In case you want to see the difference of the values one or more cuves have at different points of time, you can click on the *Show differences* button. It is located immediately at right of the *show data* button, and carries a sign “-” on it. You will get a second vertical bar such as in the two images below. Whenever you move the grey bar you get actual values; whenever you move the green one, you will get the diffe rences (difference values are marked with an asterisk “\*” at the left side of the numbers). As before, in case a single plot is shown the numbers are shown in the Plot window, otherwise in a separatate, small additional window.

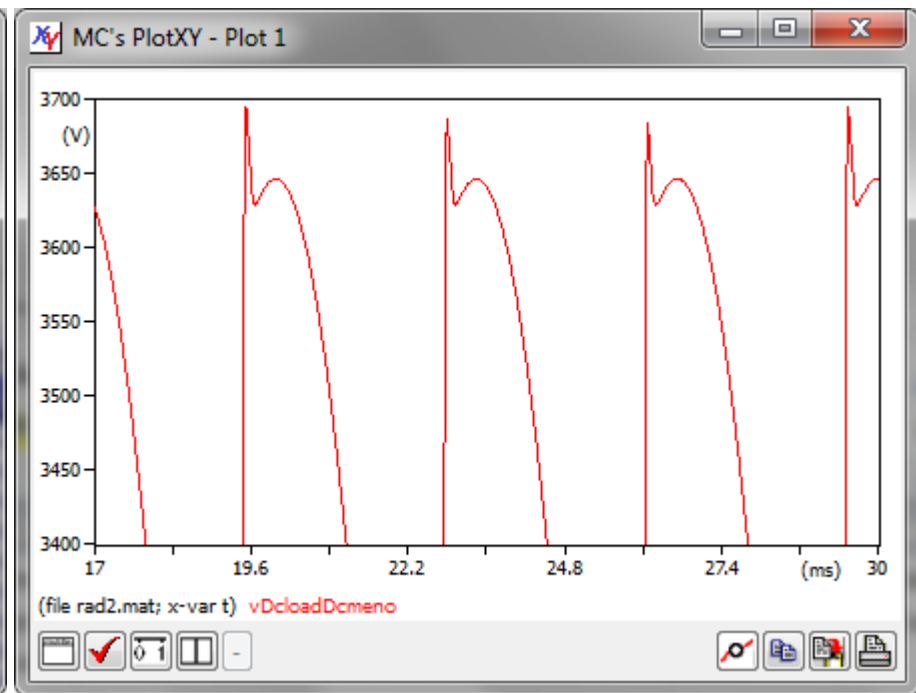
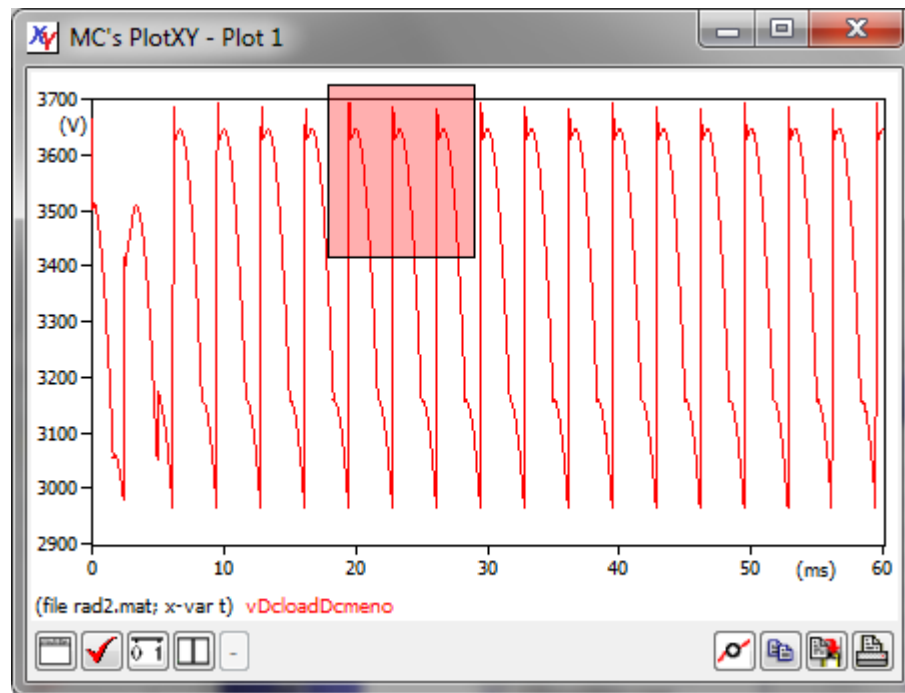




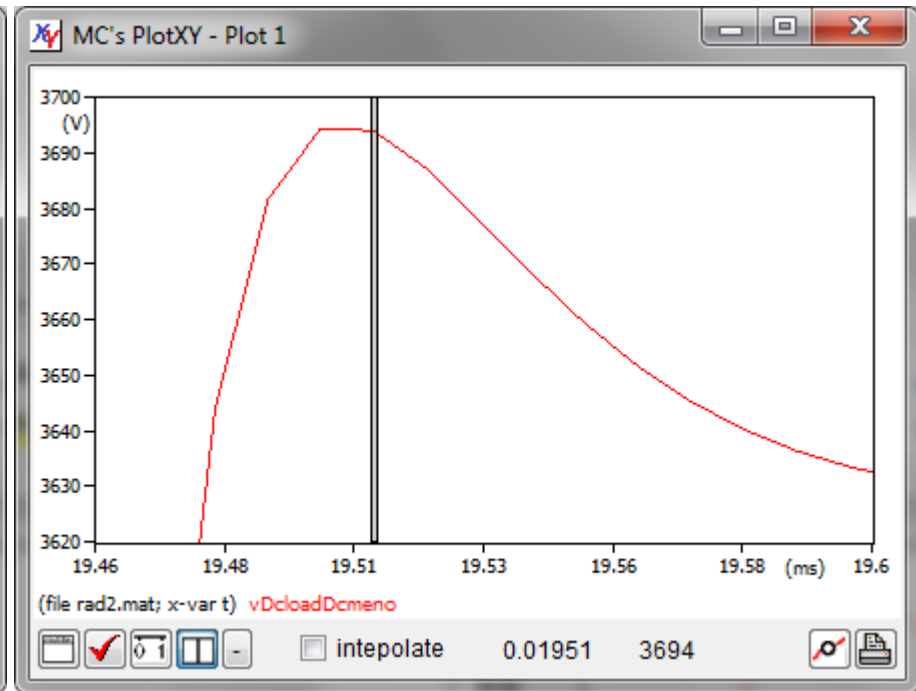
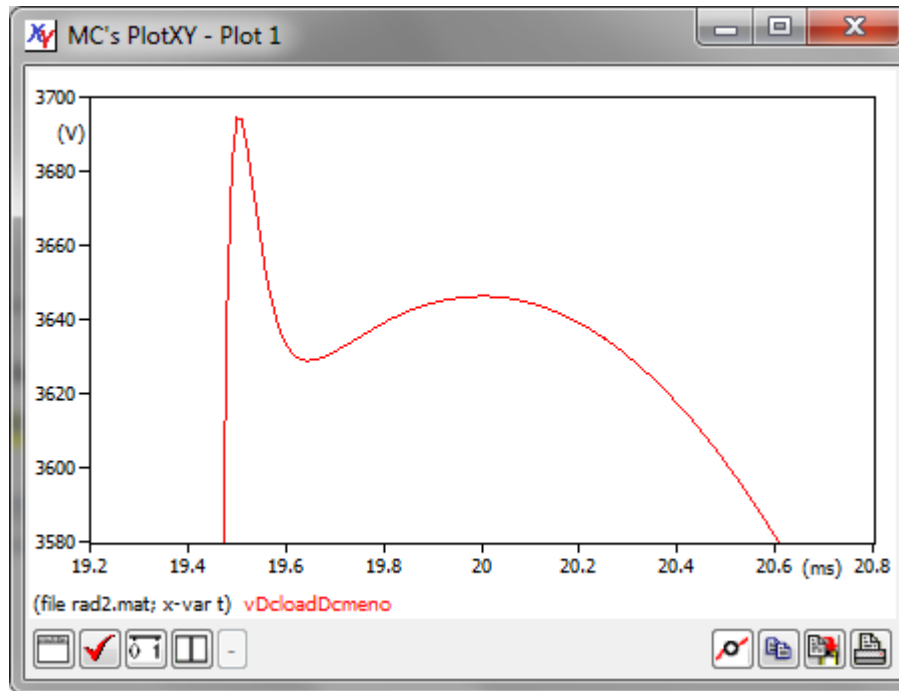
## Zooming and keeping zoomed

Very often we need to have a closer look at some plot detail. To do this we want to zoom the plot

This is straightforward with PlotXY: simply we define the zooming rectangle with the mouse: we push down the left button in the top-left corner of the rectangle, drag the mouse, and leave the button once the rectangle displays what we want. While dragging the selected area is in partly transparent pink that shows what we are selecting. See the two pictures below: the left one shows zooming in action, the right one shows the zoomed plot. Try to repeat this using the supplied file rad2.mat.



We can repeat this many times, to have a look at even tiny details of our plots. See below.



In a zoomed plot we still can look at the numerical values corresponding to the plot, either focusing on the actual data from the loaded file, or on an interpolation between points (see the right picture above).

Whenever a plot is zoomed, the axes are kept clean: we have “round numbers as minimum and maximum values and on the tic-marks. Only when the zoom is very deep, PloXY gives up choosing clean axes and use whatever it can.

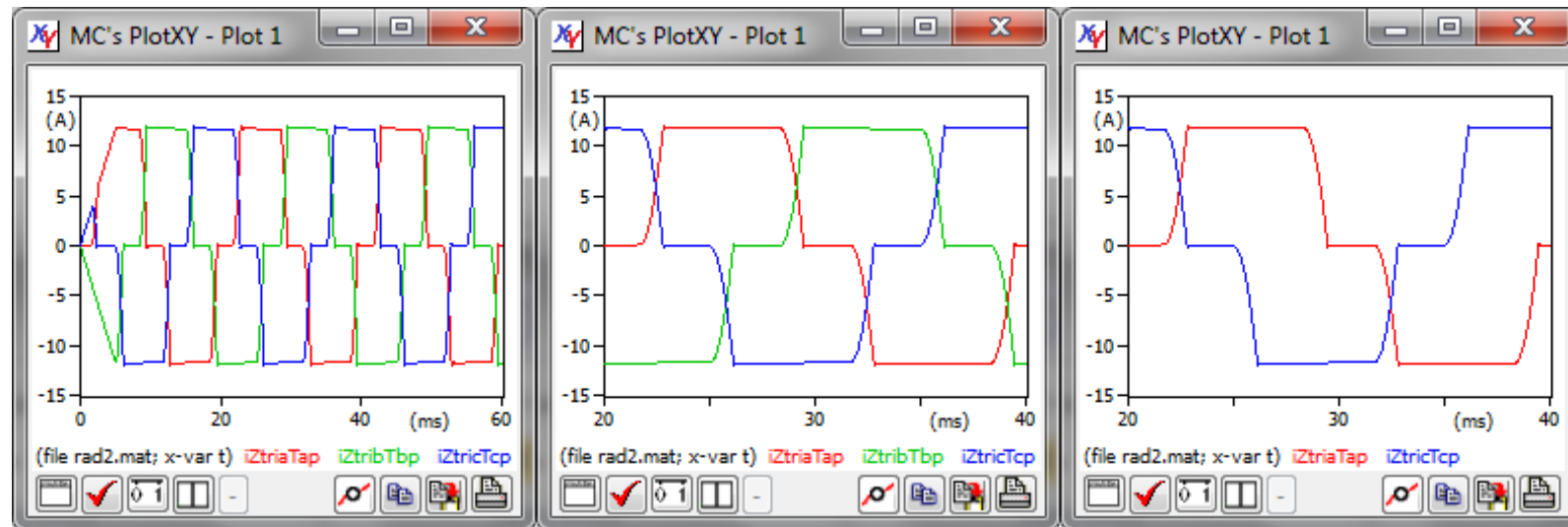
To zoom out, right-click on the plot area: we will have the choice between getting back one level or zoom out the plot completely.

It might also be useful, in some occasions, to add or remove plots from a plot window without updating the zoom range.

Let us do together an exercise. From rad2.mat we can plot variables 16, 17 and 18. The plot will appear as shown below at left. Then manually zoom between 20 and 40 ms. The resulting plot is shown below at centre.

We might now add or remove curves while leaving the axes range unchanged (between 20 and 40 ms horizontally, and between – 15 and 15 A).

This is done by clicking on the *Keep scales ranges while plotting* button (on the main buttonbox of the DataSelection Window) before clicking on the plot button. If that button is down, the axis scales are left unchanged. Therefore if for instance we remove variable N. 17, and then we plot keeping scales, the plot is as shown at the rightmost picture below.



### Special plots: with twin vertical axes or X-Y plots

Rather often the best visualisation is when two different plots are built using the same horizontal scale but they have different vertical sizes. In these cases it may be very convenient to realize plot with two different vertical axes (here called twin vertical axes).

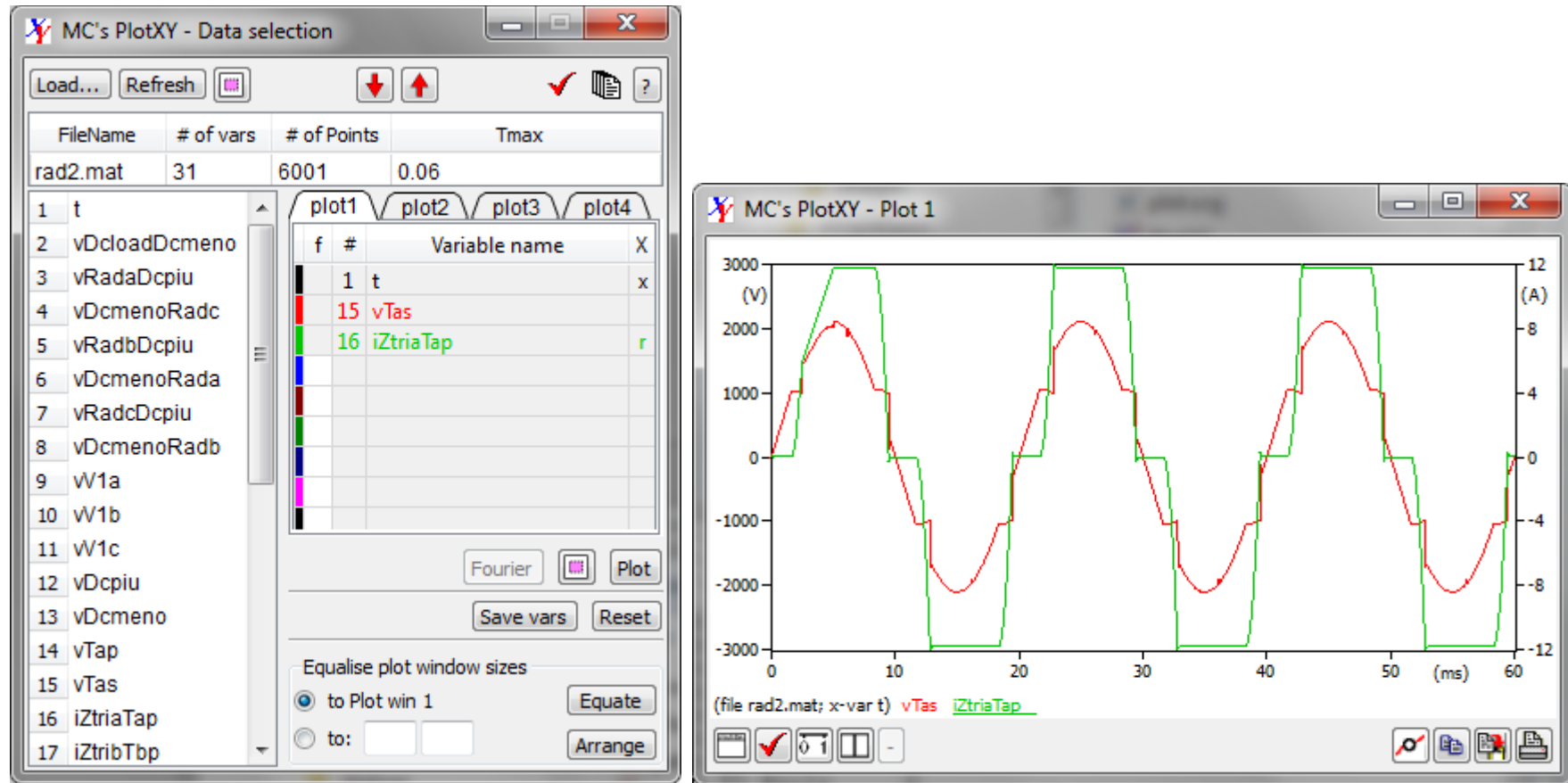
PlotXY allows you to create plots with several curves, while allowing you to select which one corresponds to the left vertical axis and which correspond to the right one.

This can be seen using the enclosed file sample2.mat.

The figures below show what happens if we select a voltage (variable vTas) and a current (iZtriaTap) as variables to be numerically evaluated against the left and right vertical axes respectively.

The first variable has been selected by left-clicking on its name, the second one right-clicking on its one. In the plot window we can easily understand with variables refer to the right vertical axis: their name is shown underlined.

*Note:* You can also right-click on the name of an already selected variable to toggle between left and right vertical axis usage for that variable.

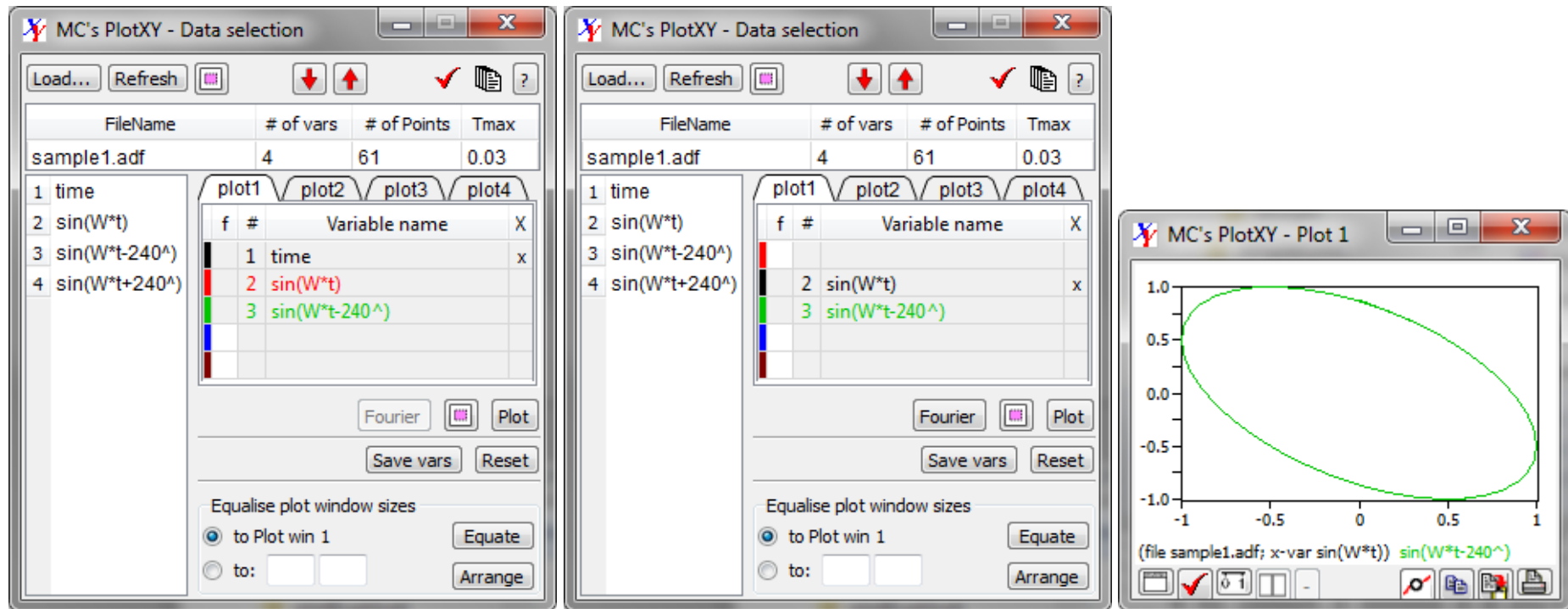


Another useful feature is the possibility to plot a variable *against* another.

This can be done only from SelectdVars tables were the selected variables come all from the same file. To make X-Y plots:

1. select the variables of interest and
2. click on the "X" column in tie plot var-list of the *dataSelection* window.

When step 2 is performed, the row onto which one has clicked becomes the “x” variable for X-Y plots. When this step 2 is performed for the first time after the latest reset of the Plot var-list table the previous x-var is removed from the table, assuming that it represents time, and therefore it is not wished. In case of subsequent clicks, however, the old X variable is not automatically removed and becomes one of the y variables of the X-Y plot.

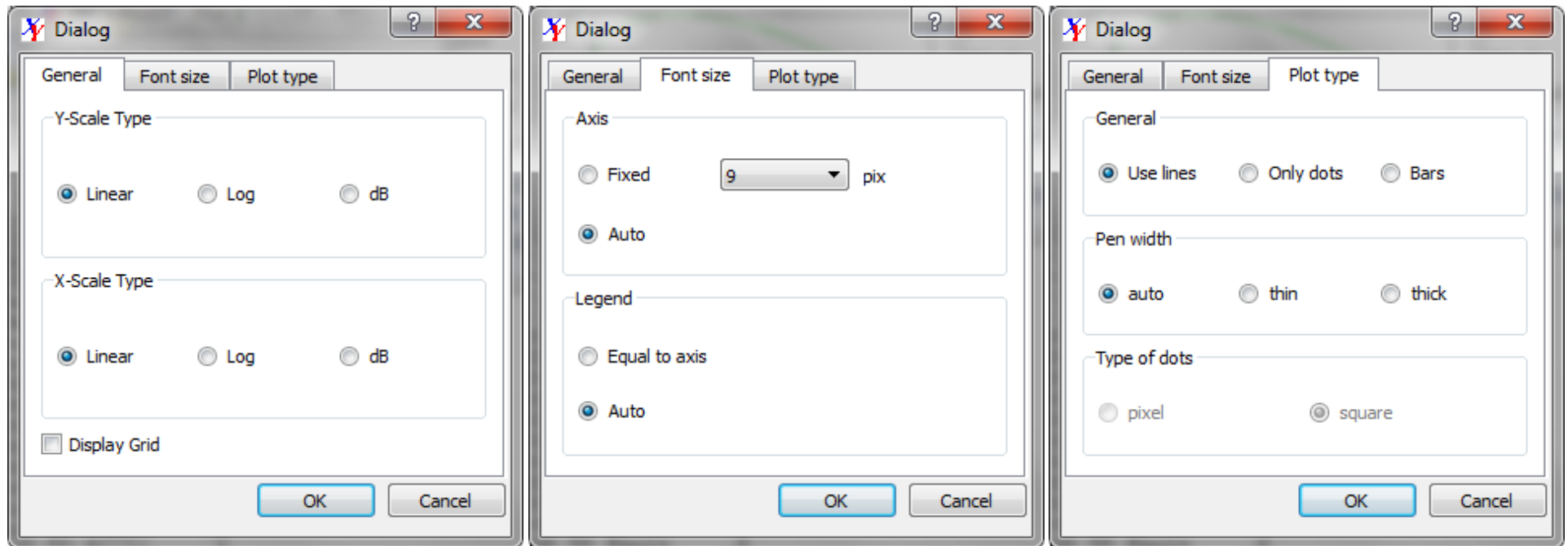


## Changing the plot type and appearance

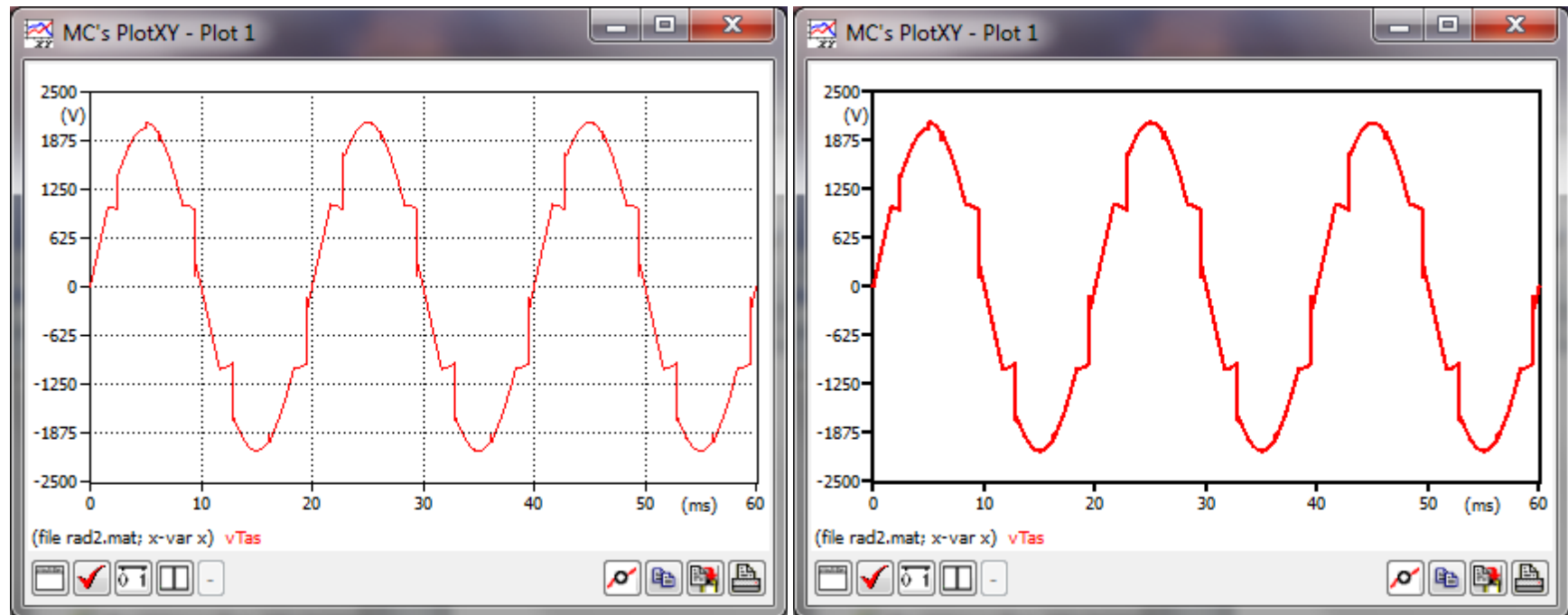
In the previous section line plots with linear scale on the axes were shown.

Naturally there exist several options to change the appearance of the plot. To explore them, try to click on the *Plot options* or *Change scale* buttons.

Click for instance the *Plot options* button. A window like the ones shown below appears. The window has three tabs displaying different options each. The meaning of the options is self-explanatory.



Two examples of the application of some plot options are shown below: General | Display Grid (left), Plot type | pen width | thick and Font size | axis | fixed 12 pts (right).

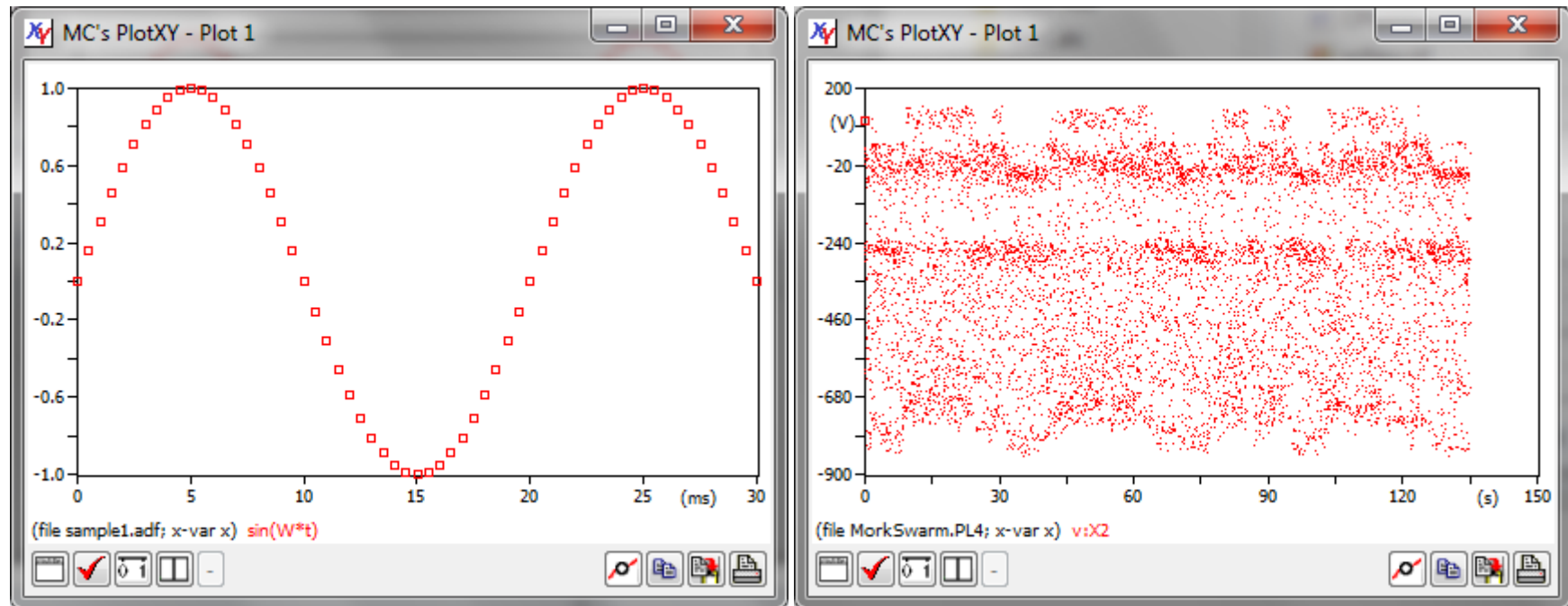


When you use a log axis, the marks on the axes and gridlines will be at 2x, 4x, 8x the main power of ten.

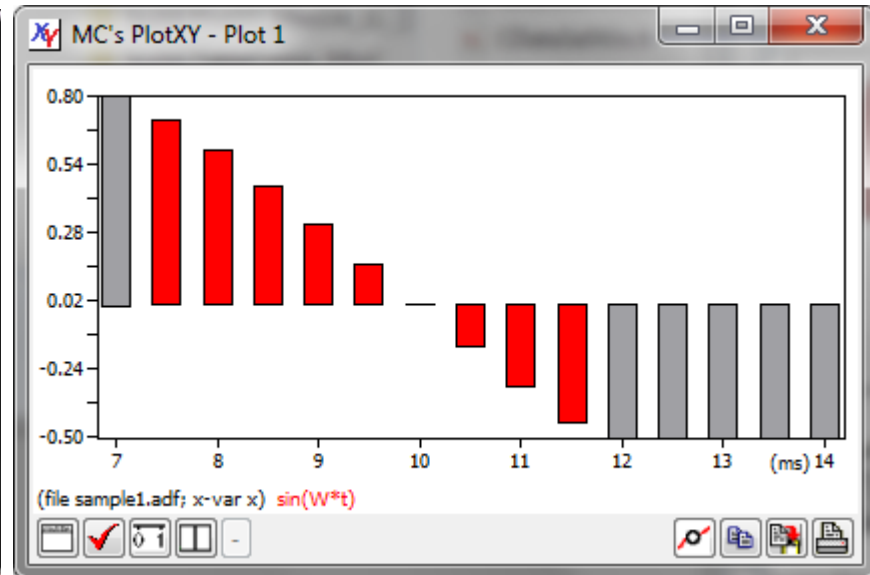
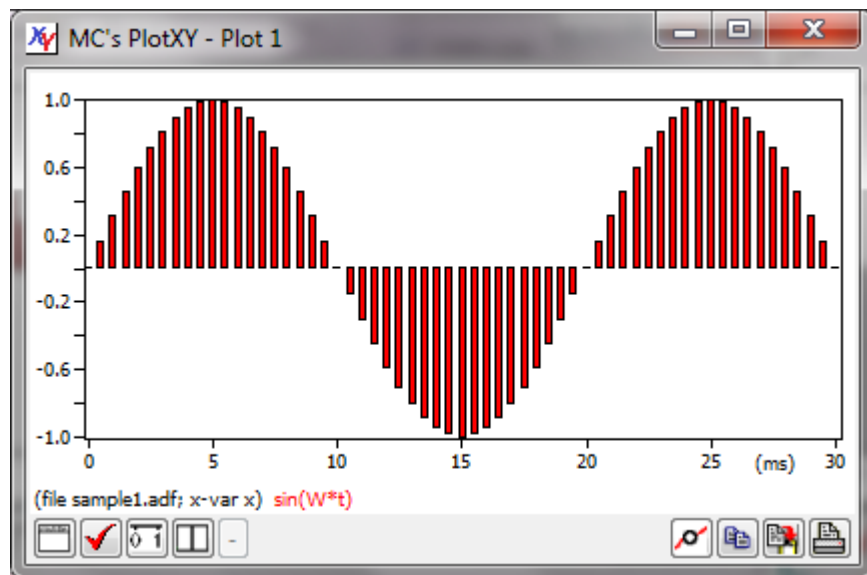
There are cases in which line plots are not the best way to show a plot. If we want to see only the point read from file we can use *General/only dots* option. This is done in the bottom-left plot for the first variable from sample1.adf file.

Finally there are files in which the X-Y correlation is weak; this can be shown using a cloud of points. In this case small squares to show points are too large, and it is better to use single pixels. This is done in the bottom-right plot, using variable X2 from the MorkSwarm.pl4 file

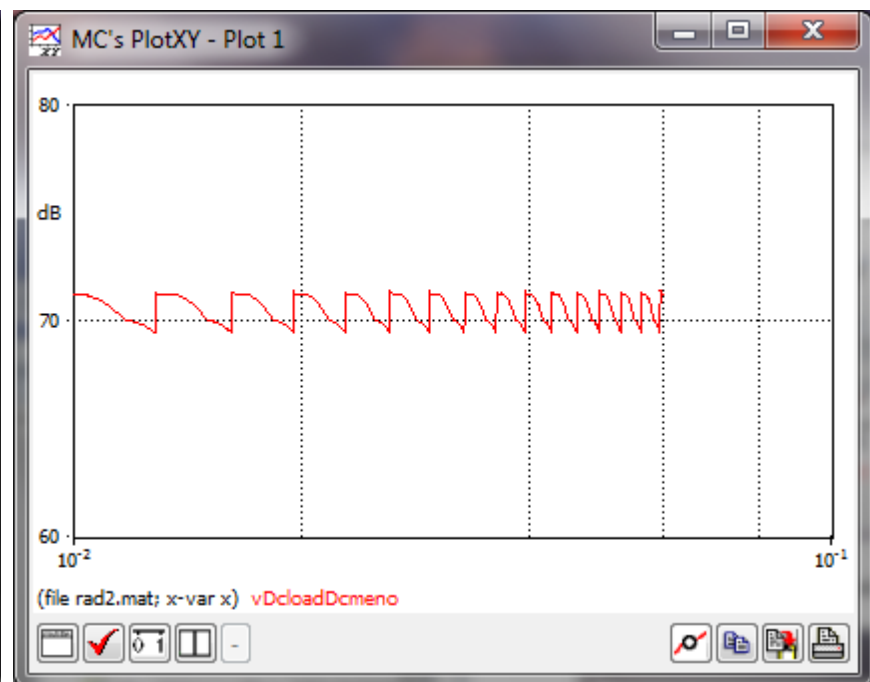
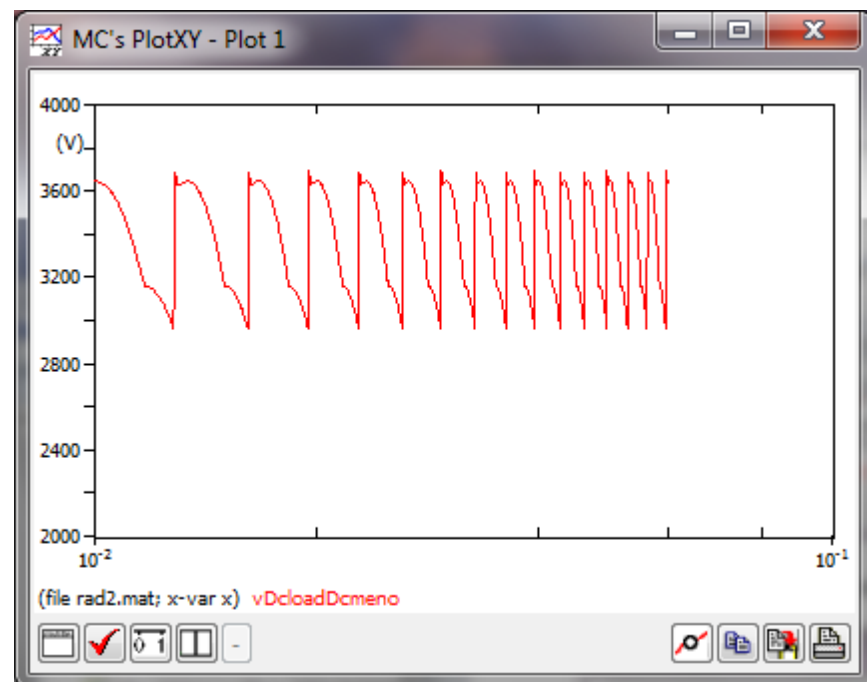




There are also cases in which instead of line plots the user needs to use a bar chart. This is done in the bottom-left plot (from sample1.df) in full scale, and in the bottom-right in a zoomed fashion. The bars whose second end falls outside the zooming window are shown greyed..



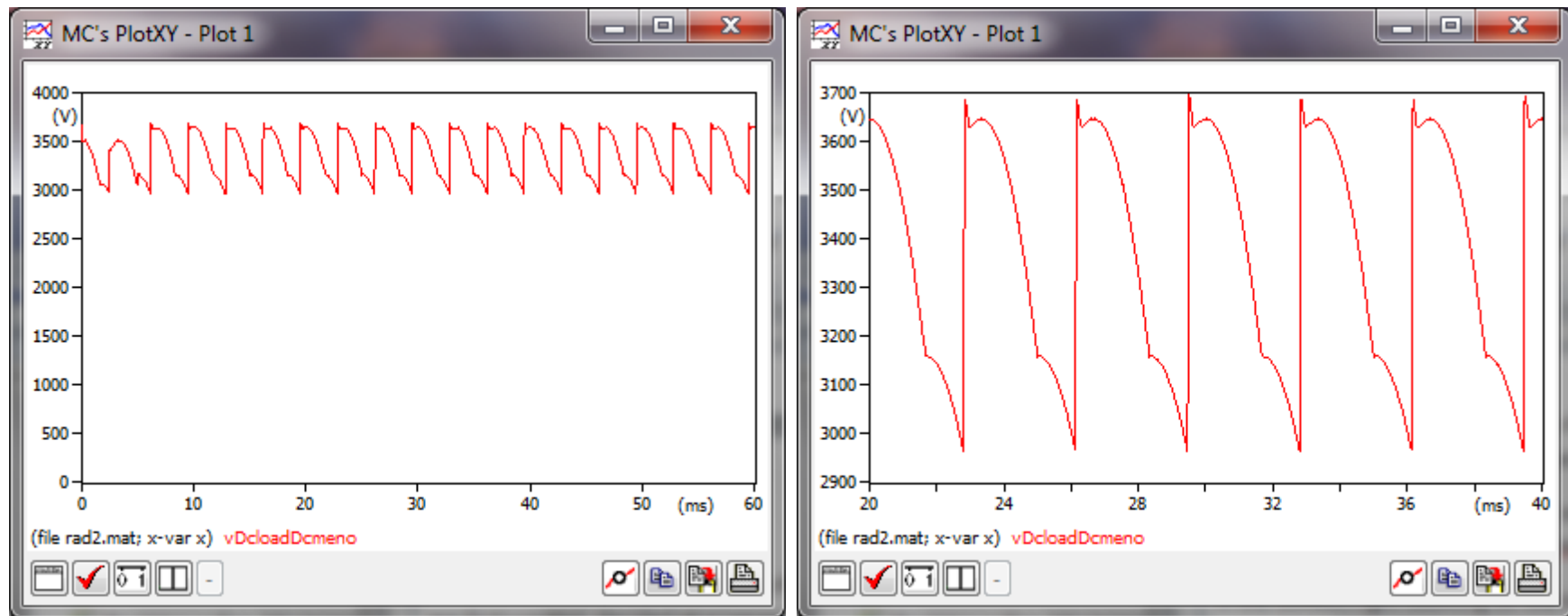
Finally scales can be linear, log, and dB. Two examples from sample2.mat are shown below



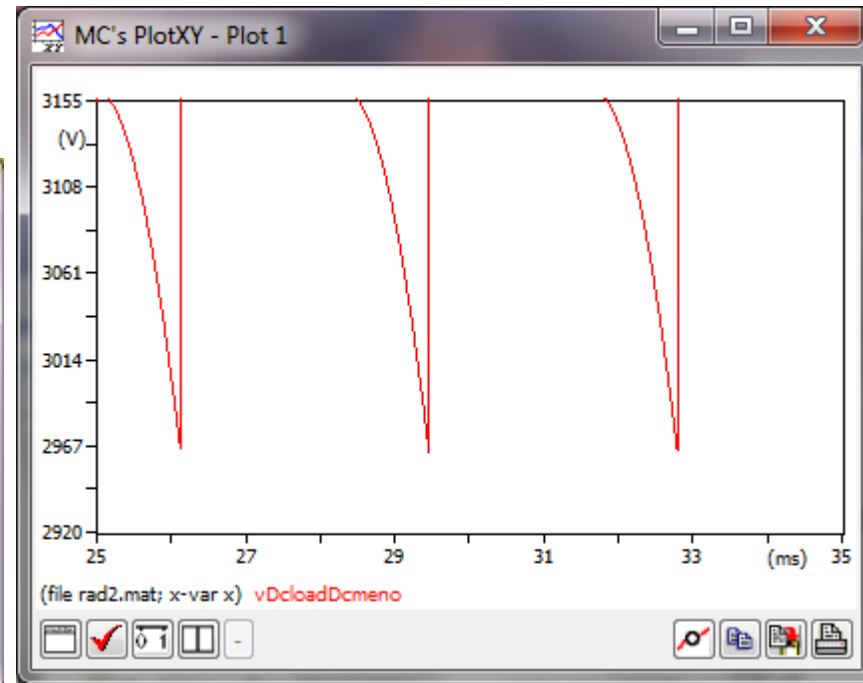
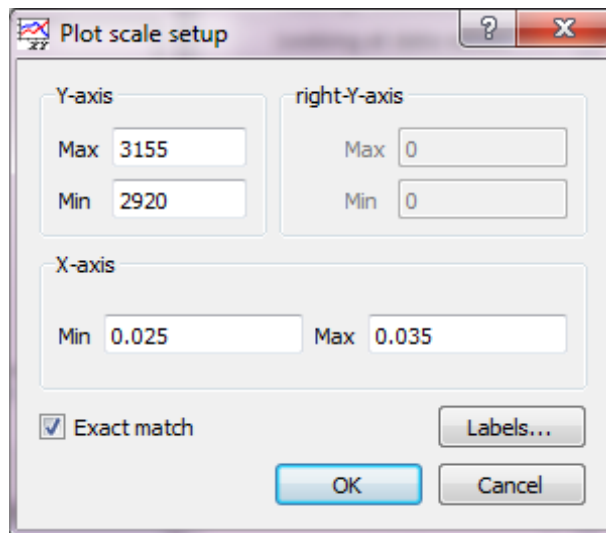
The customisation you do using the *Plot Options* button are valid for the current plot. The default graphical characteristics of the plots, e.g. those valid when plots are first created, decided at program level, using the *Program options* button of the program main window.

The Change scale button allows to manually change axes scales: this allows more flexibility than zooming: you can pan or choose detailed scales. In the bottom-left plot the variable vDcLoadDcmemo is shown between 0 and 400 V, overriding the default scales.

In the bottom-right zooming on the horizontal axis is made.



Normally the user selected scales in the *Plot scale setup* dialog are subject to some refining by the scale determination inner engine to allow easy numbers on the numerical tic marks. If, however the user wants total freedom in choosing minimum and maximum values of an axis, he can attain this checking the Exact match checkbox. See the example below.



## Exporting selected variables into a new data file

Very often files containing output from simulations or measure contain a lot of variables, while only a few of them are of interest for a given purpose.

PlotXY is able to create new files with such a subset of variables. Currently, however, saving only can be made from variables coming from the same input file. Thus, variables from different files or variables created using [The "function of variables" feature](#), cannot be saved.

To export the selected variables into a new data file click on the *Save vars* button of the in the *Data Selection* window.

Note that pl4 as output file type is only allowed when saving variables already belonging to a pl4 file. In this case the pl4 file created is of the so-called "pisa" format (or newpl4=2 format).

## Exporting plot into system clipboard or SVG/PNG

Once you are satisfied with your plot you may want to export it somewhere.

Maybe you want to include it into a document file (e.g. Microsoft Word or OpenOffice writer): to do this you click on the *Copy* button on a Plot window, and then paste it into your favourite program. This way you get a pixmap copy (once called bitmap copy) of the plot.

As an alternative you can save your plots onto disk. In this case, you can obtain this result clicking, still in the plot window, on the *Save plot* button. You will get both a Portable Network Graphic (“PNG”) and a scalable Vector Graphic (“SVG”) plot copies on your hard disk. The PNG is a pix map while SVG is a vector copy. The SVG has the advantage that it can be manipulated using a SVG editing program (such as the freely available *Inkscape* from [www.inkscape.org](http://www.inkscape.org) or the commercial Microsoft’s Visio). In this way you will find that curves will remain curves (can be moved around as a single object, text items will remain text items, etc. Moreover zooming can be done without loss of visual quality.

Note that another way to have a picture on file of your plots is to use the [Printing on paper or pdf](#) feature.

### The “function of variables” feature

PlotXY not only allows power display of the contents of your variables, but also can manipulate your variables to obtain new ones, and plot them. You can for instance sum, multiply, divide your variables. Parentheses are allowed.

You can also request the time integral of a variable or an expression. In this case, the string you enter to define the function of variable must begin with “int” and end with “)”

*NOTE. In the current version of PlotXY, functions of variables can be used only when all the variables involved in a function come from the same input file. “Time”, used to compute the time integrals is assumed to be the first variable of the file the function of variable to be integrated refers to.*

The newly created variables are called functions of variables.

To define functions of variables you access your variables by a *compact name* built according to the following conventions:

- In general the compact name indicates the file number and variable number with the scheme “f#v#”. For instance variable number 3 from file number 5 has as compact name “f3v5”
- To indicate Variables from the *Selected file* (that is indicated by a “x” in the first column of the *File list* table) the file indication can be dropped. For instance the 5<sup>th</sup> variable from the Selected file can just be indicated as “v5”.

Functions of variables are defined by clicking on a cell in the “f” column of the *File list table* in the *Data selection* window. A small window will appear prompting you to introduce the string defining your function of variable.

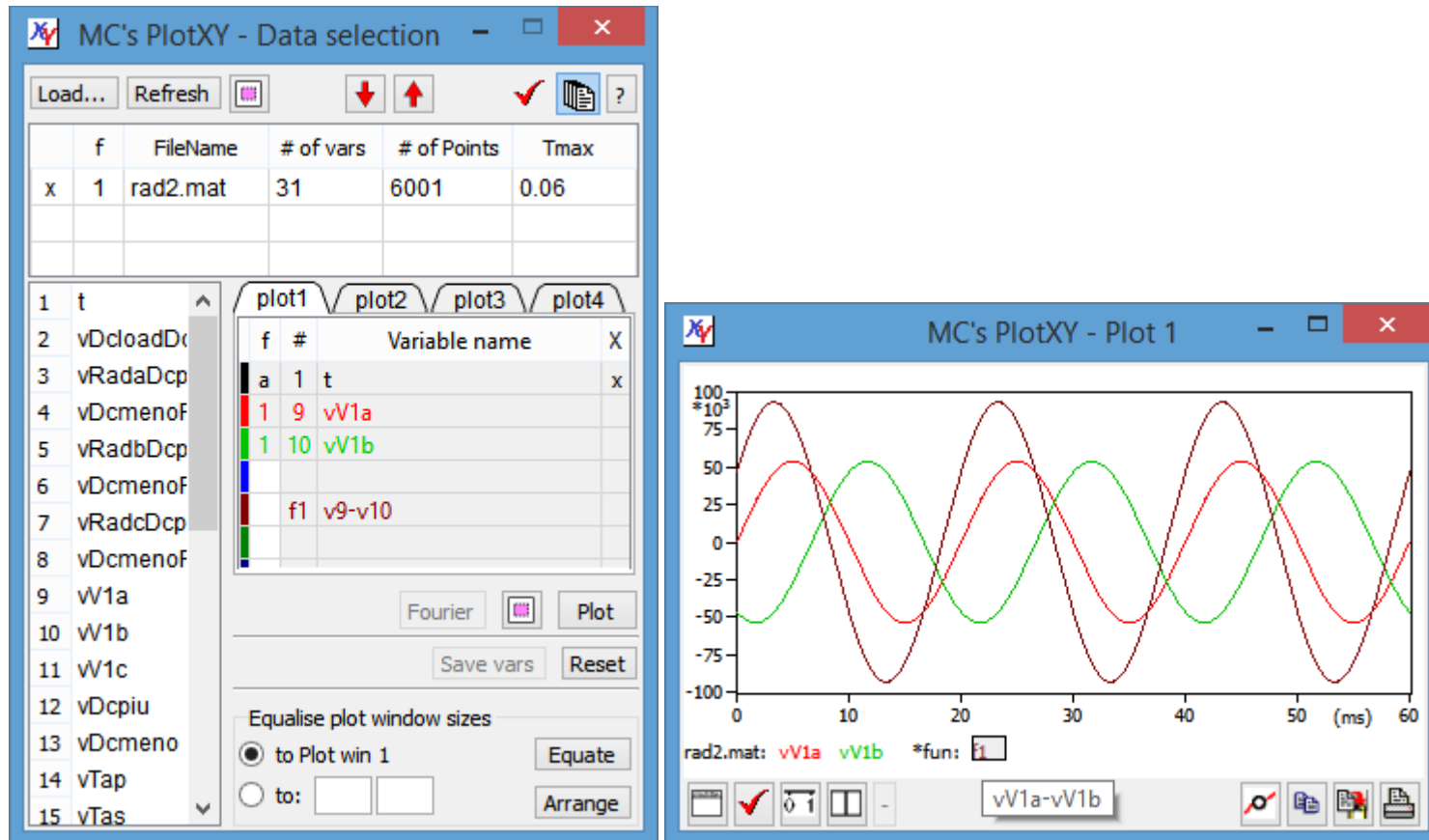
Valid strings could for instance be: “v9-v10”, “f1v1+f1v2+f1v3”, “v1-(v2+v3)”, “10\*v9+20”, “v1” “int(v2+v3)”,. The latter example request the integral between t=0 and the generic time t of v2+v3.

The first and last strings are used in the examples are shown below. The integral is shown in the same plot along with the function to be integrated for the maximum clarity.

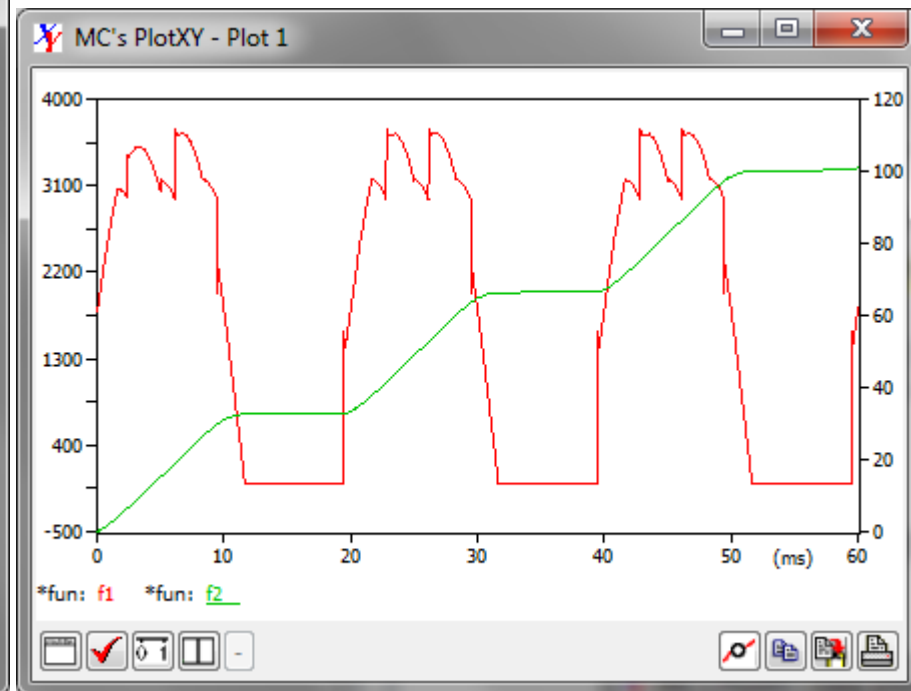
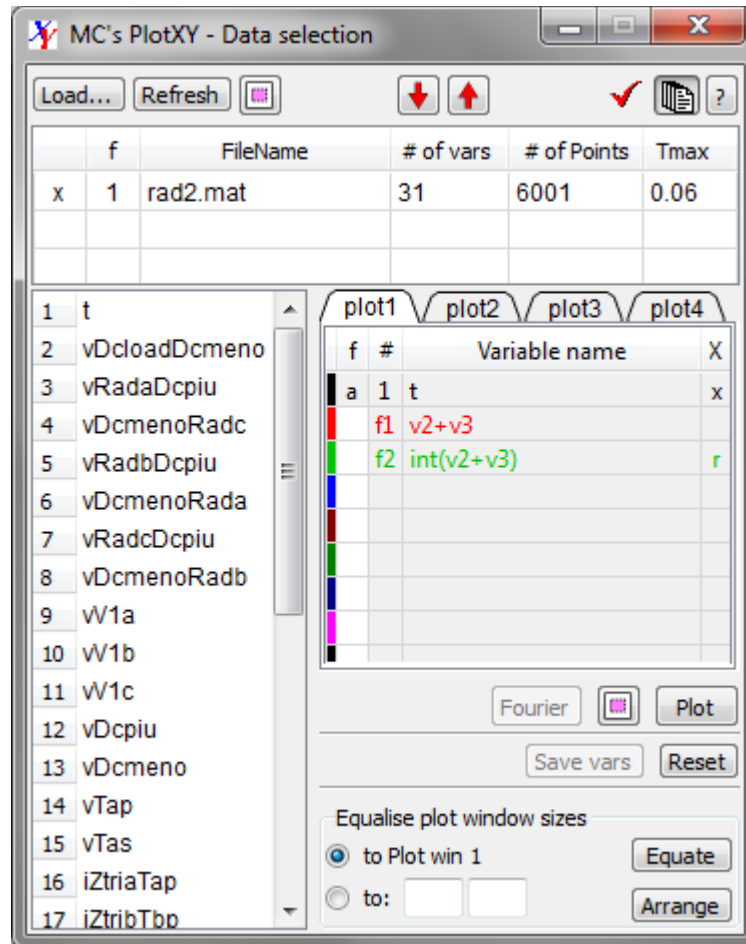
NOTE1 These compact names are created especially thinking of future versions of PlotXY in which one will be able to mix variables from different files. By now, however, only functions for which all variables belong to the same file are allowed, and therefore the “f#” part of the name is nearly always useless. Indeed, in current version, the program automatically removes the redundant “f#” part of variable names.

NOTE2. The name of the “function of variable” is just indicated as “f1”, “f2”, etc. However, to ease analysis, if the mouse is left above the variable name, the full representation of the used formula is shown.

For instance, in the picture right below the mouse is left on “f1”, the user will see in a tooltip: “vV1a-vV1b”

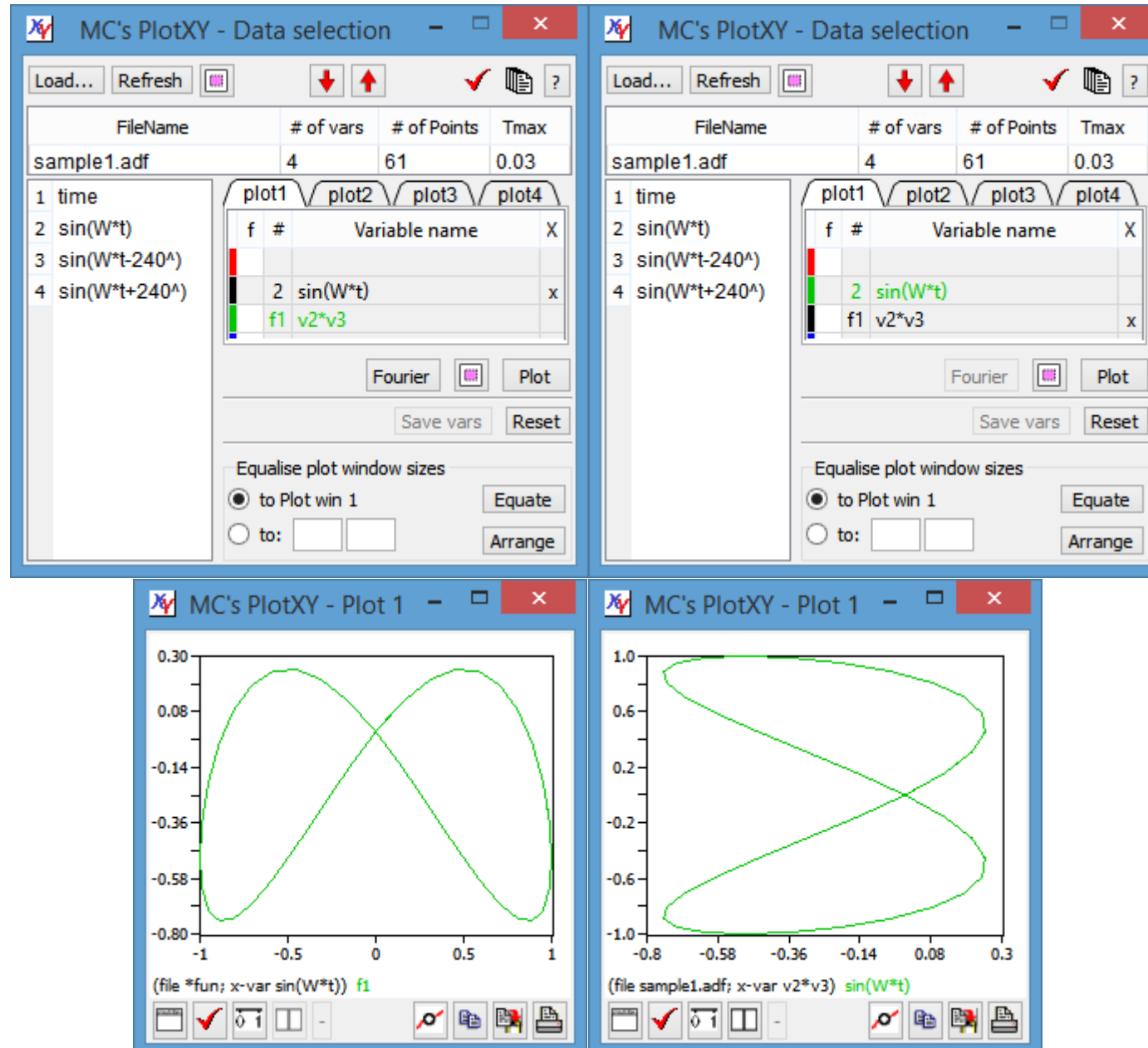


Note that If you hold the mouse pointer onto the function name in the plot window you get a full indication of what that function means (here “vV1a-vV1b”)



NOTE: integrals are useful to calculate energy flows. For instance  $\text{int}(f \cdot v)$  and  $\text{int}(u \cdot i)$  will calculate the mechanical energy and electrical energy if  $f$ ,  $v$ ,  $u$ ,  $i$  stand for force, speed, voltage, current respectively.

Finally consider that also the x variable can be a function of variable. Compare the two plots below:





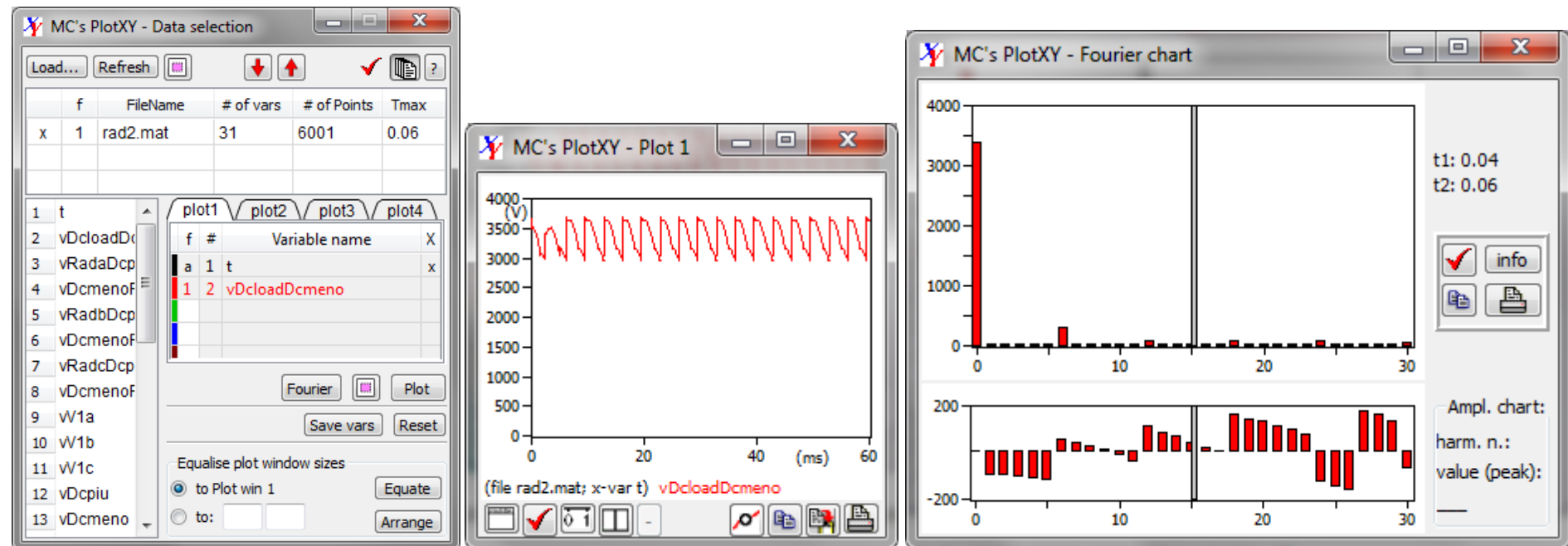
## Creating a Fourier Plot

Electric and sound engineers use a lot Fourier decomposition of periodic waves.

This is well supported by PlotXY.

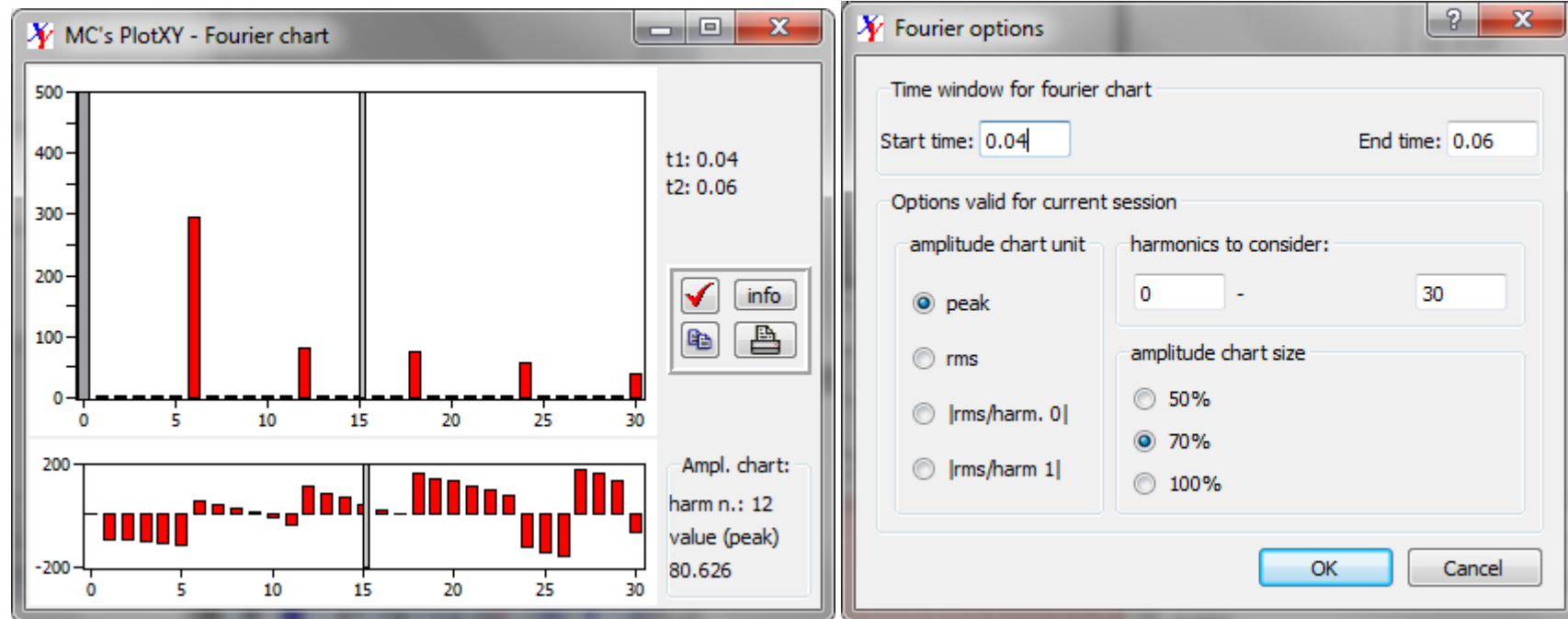
To make a Fourier plot only a unique variable must be selected. Then, simply click on the “Four” button. The program will try to choose the best parameters for general usage. In particular it assumes that the simulation, after a possible initial transient has stabilised. Therefore, using the default frequency (that can be selected by means of the Plot Button of the DataSelection Window) it uses the last period of the available data to make computations.

Take for instance the following pictures, obtained using the supplied rad2.mat file. Here the default frequency is 50 Hz and the simulation goes from zero to 0.06 s, it will use as default the last twenty-millisecond period, for the analysis



Once the chart is shown, you can move around the data cursors to see numerical values in the bottom -right part of the window.

You can zoom the chart. An example is shown below (on the left). The bars that fall into the plot are shown as usually in red (bounded by black rectangles), while the bars that are higher than the available vertical space are shown greyed.

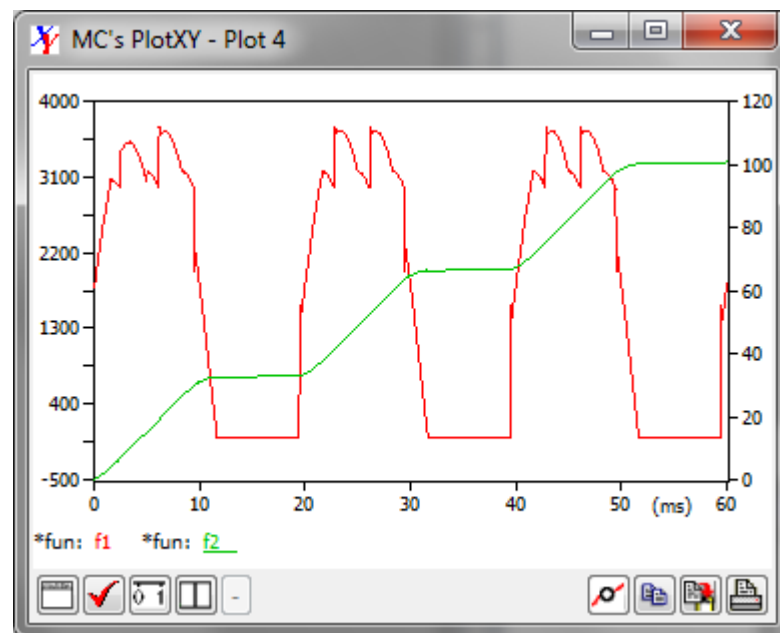
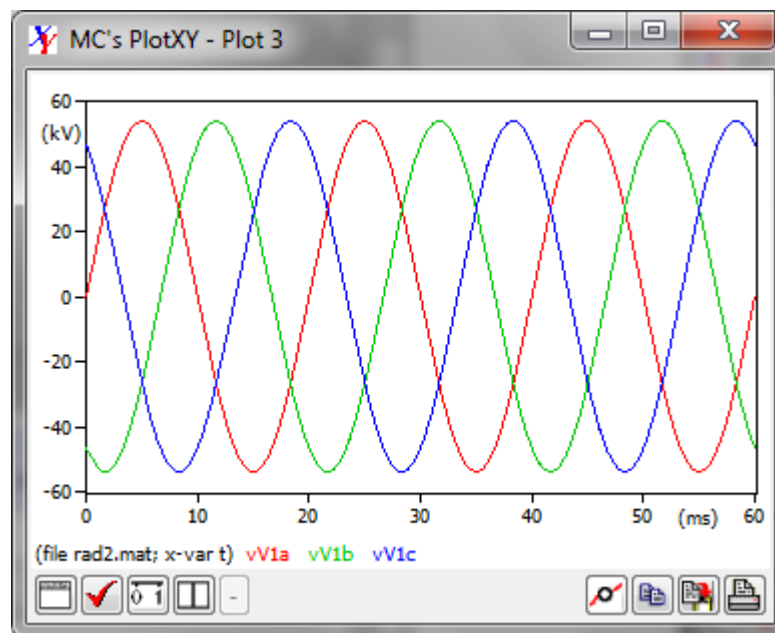
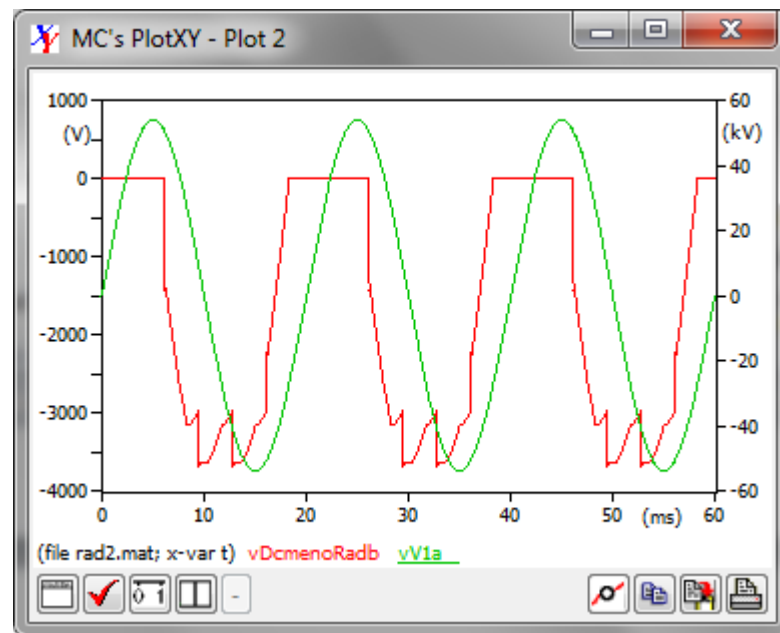
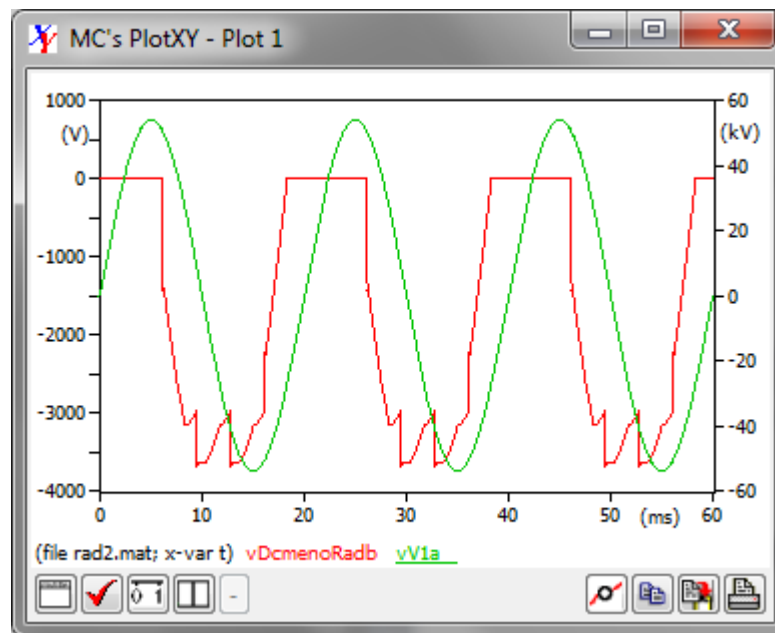


The fourier Chart can be customised in several ways clicking on the Fourier Options button. The window that appears has the aspect shown above (on the right). In particular note that the numerical values can be shown in the Fourier Chart window in four different values:

- peak of sine components (DC component unchanged)
- rms of sine components (DC component unchanged)
- all harmonics as a ratio to the DC component value
- all harmonics as a ratio to the first harmonic value.

### Using several program windows

Starting from the same files and variables you can shoe plots in up to four windows. Each window has its own setups, line style, scale, data-browse window, etc. Example are shown below.



## Saving Program State

When we work on PlotXY we load files, select variables, define functions, make plots.

When we close it, all this setup is in general lost. To avoid this, one can “save Program State” , i.e. the list of files loaded, variables and functions per file and plots shown, so that he can reload it again at a later time. The state is automatically stored in the Operating System’s program setting area (in Windows this is the Windows registry) and can be recalled at will.

To save the program’s state click on the “Save state” button on the main toolbar of The DataSelection Window.

Note that when working with PlotXY it may happen that a plot shows contents that does not correspond to the corresponding selectedVarTable. For instance we could select in the Plot1 selectedVarTable variables X and Y; plot them, and select a different set of variables without hitting the plot button again.

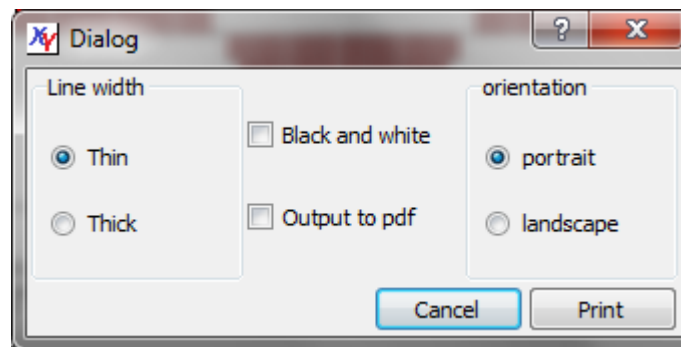
When the program state is restored (clicking on the Load state on the Data Selection window), however plots always correspond to the corresponding selectedVarTable contents, as if the user has just hit the plot button.

NOTES:

- Fourier plots are not automatically generated when restoring the program state.
- The plot’s appearance (line vs bars vs dots, zoom state) is not stored.

## Printing on paper or pdf

Plots can be printed on paper or pdf. The options at your disposal are those shown in the following diagram, that appears when you click on the “print” button of the plot window.



### **Automatic loading of files.**

It may happen that you always (or very often) load the same files or files having the same name

In this case, in addition to the “save state” feature you have another option: specifying the files at command line.

If, for instance, in Microsoft windows you can create a link to PlotXY.exe in your folder containing “sample1.adf” and “rad2.mat” and writing in the program name after “PlotXY.exe” (without quotes) “rad2.mat sample1.adf” (without quotes). Your PlotXY will open and automatically load the two files.

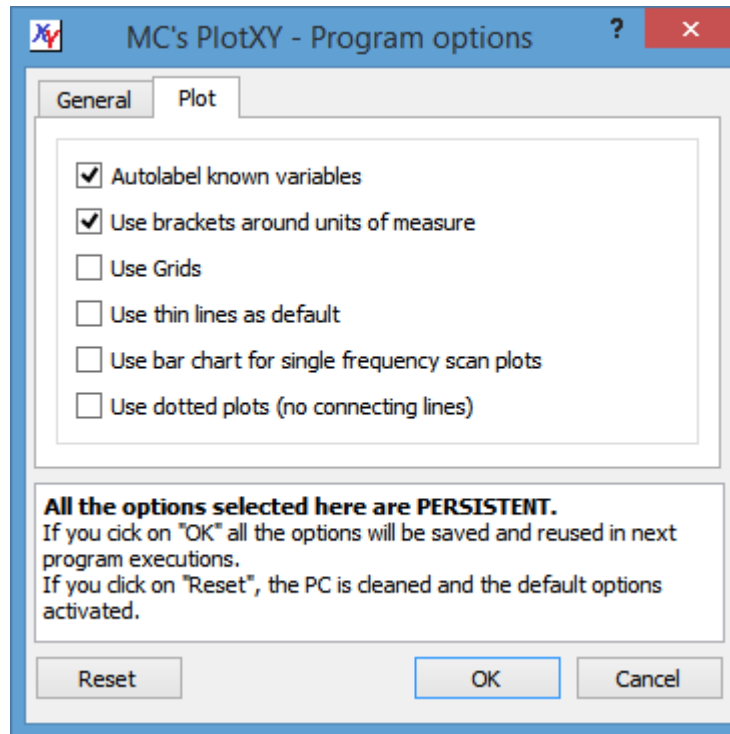
You can list up to eight files as command-line parameters. They all will automatically be loaded.

### **Additional information**

#### **Customizing the program behaviour**

We have already met in section named “Loading a file and doing first things” the “Program Options window.

There we saw the appearance of the “General” tab sheet. We also have other options to choose from in tab sheet named “Plot”, shat is shown below: here.



Their meaning is rather straightforward. Only the “autolabel known variables” might need some explanation and this is done in the following “Automatic Units of measure and prefix” section.

### Automatic Units of measure and prefix

You may have noticed that in some plots units of measure (such as (V) or (kV)) have appeared.

This by default is automatically done by the program based on the following simple rules:

The “prefix”, substitutes the powers of ten, according to the standard notation, shown for completeness in the following table:

| <i>prefix</i> | <i>meaning</i>    | <i>prefix</i> | <i>meaning</i>   |
|---------------|-------------------|---------------|------------------|
| p             | $\times 10^{-12}$ | k             | $\times 10^3$    |
| n             | $\times 10^{-9}$  | M             | $\times 10^6$    |
| u             | $\times 10^{-6}$  | G             | $\times 10^9$    |
| m             | $\times 10^{-3}$  | T             | $\times 10^{12}$ |

The units of measure are chosen as a function of the first character of the variable name, according to the following table:

| <i>name begins with</i> | <i>assumed unit</i> | <i>symbol</i> | <i>name begins with</i> | <i>assumed unit</i> | <i>symbol</i> |
|-------------------------|---------------------|---------------|-------------------------|---------------------|---------------|
| 'v'                     | volt                | V             |                         |                     |               |
| 'c'                     | ampere              | A             | 'i'                     | ampere              | A             |
| 'p'                     | power               | W             | e                       | energy              | J             |
| 't'                     | time                | s             | 'f'                     | frequency           | Hz            |

Note that the last correspondence is used for horizontal-axis variables only

You can disable this feature unchecking the “autolabel known variables” check box in the Program Options | Plot tab sheet (cf. section [Customizing the program behaviour](#))

## More on loading files

In section Loading a file and doing first things it was said that loading files can be done either using the Load... button, or dragging files on the Data Selection window. Here I supply some more details.

With both techniques you can load individual files or groups of files. The program will adapt to different situations and behave as expected. At the end of load operation more than one file is loaded, the program is in multi-file mode (it goes into this mode if it was in single-file mode).

However, when individual files are loaded starting from single-file mode, the program behaviour depends on whether the action is initiated from single-file or multi-file mode:

- when starting from multi-file mode the new loaded or dragged file is added to the already loaded ones
- when starting from single-file mode, the new loaded or dragged tile replace the currently displayed one.

I recommend to try these differences: once you get accustomed, you can take advantage of them.

## What else?

In this tutorial you have learnt all the major things you can do with PlotXY. You can learn the rest just by trying.

However, there are specific details on the structure of adf file or the rules that the program use to convert variable names into the supplied formats: for instance not all the names that are valid as ADF file or pl4 file names can be used as such in matlab files.

This more advanced documentation is provided document: “Input formats and naming conventions.pdf”

