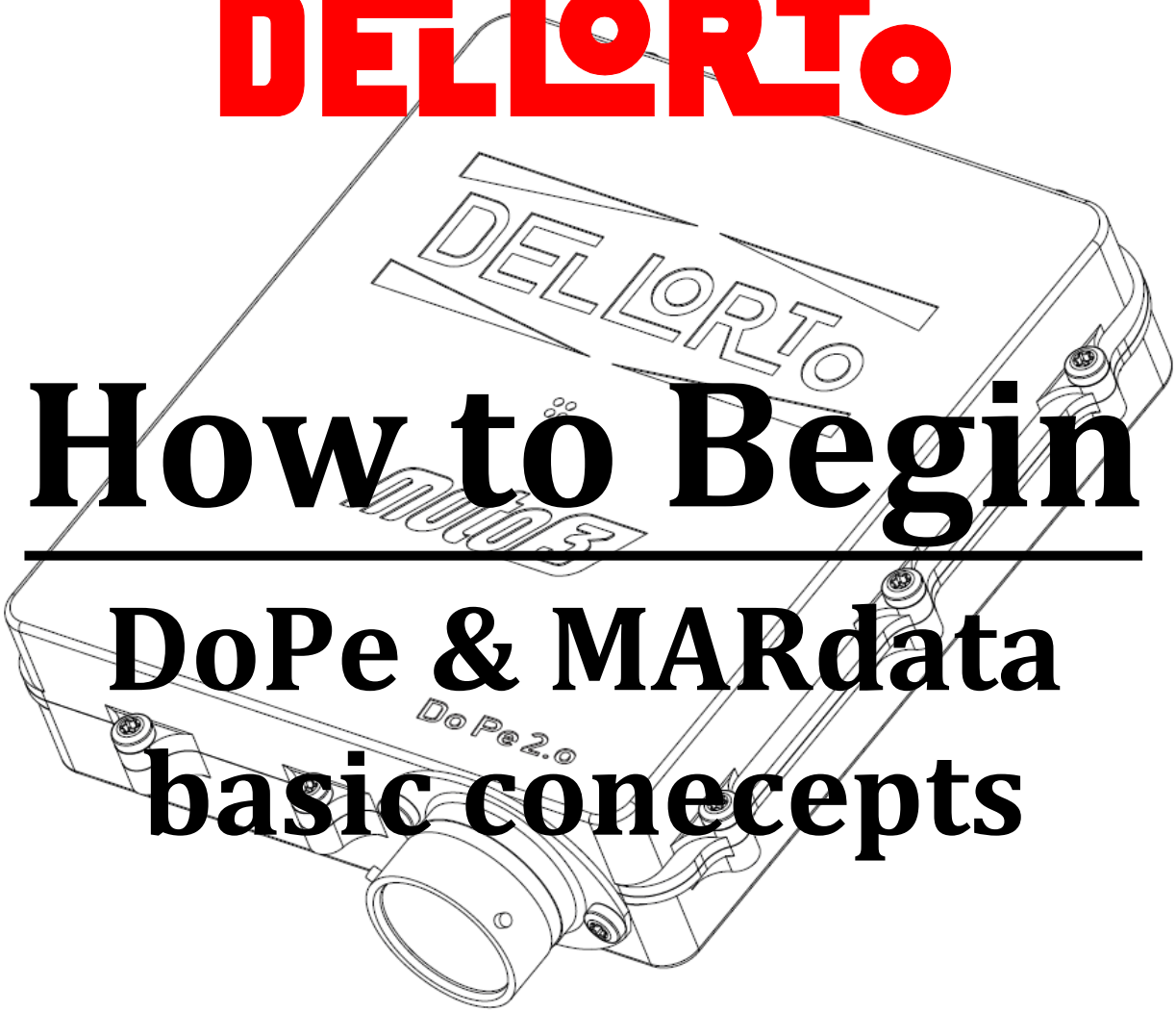
DELLORTO



How to Begin

DoPe & MARdata basic conecepts

moto3

Version V2

March 2015

DELL'ORTO S.p.A.

Sede Legale:
Via R. Sanzio, 2 - 20831 Seregno (MB) - Italia

Sede Operativa e Uffici:
Via Kennedy, 7 - 22060 Cabiate (CO) - Italia
T. +39 031.769211 - F. +39 031.7692216 C.P. 92
www.dellorto.it - staff@dellorto.it
http://www.dellorto.it/privacy.asp

Cap. Soc. int. vers 5.160.000€
CF 00685940157 - PI 02830910960
Registro delle imprese di Monza e Brianza n° 00685940157

1 Before installing a software 3
2 How to configure your PC to connect it to the DoPe ECU 3
3 DoPe software 4
3.1 Select the ECU 4
3.2 Rx the ECU 6
3.3 Tx the ECU 7
3.4 Configure the Throttle Position Sensor, Gear Position Sensor and DC Motor 7
3.5 How to compare two configuration 8
3.6 Saving the new configuration 10
4 MARdata software 12
4.1 Download the acquisition file from the ECU DoPe 13
4.2 How to fill in all the fields of the Data Download window: 14
4.3 How to open an acquisition file that we have already saved in the computer 20
4.4 How to add channesl to the graph. 21
4.5 Graph 22
4.6 Channels name and scale (bars) 23
4.7 Channels value list 23
4.8 ScanLine tools 25
4.9 Graph cursors 25
4.10 Zoom tools 26
4.11 Channel properties 26
4.12 How to move track finish line. 29

1 Before installing a software

If the software installation folder is compressed first you must decompress it with software such as Winzip, 7zip, Winrar, etc… and then start the installation by clicking the Setup.exe

2 How to configure your PC to connect it to the DoPe ECU

It is necessary to configure the Ethernet protocol. This operation will be necessary only once.

Windows 7

1. Power on the DoPe ECU and connect it to the PC with the Ethernet wire.
2. Click on Start menu.
3. Click on Control Panel
4. Click on Network and Sharing Center
5. Click on Local Area Connection (LAN)
6. In the window, click on Internet Protocol Version 4 (TCP/IPv4) or Internet Protocol Version (TCP/IP) and then click on Properties.
7. Select: Use the following IP address.
8. Set the IP Address as: 10.1.xxx.yyy

   where xxx must be different from 116 and 118

   yyy any number.
9. Subnet mask 255.255.0.0

ⓘIMPORTANT: If there are communication problems with the ECU DoPe, please check if your antivirus and firewall are disable.

Rarely (for example when using the McAfee antivirus software) even disabling the antivirus is not enough to solve DoPe communication problems. This cases, the only solution is to remove the antivirus software at all.

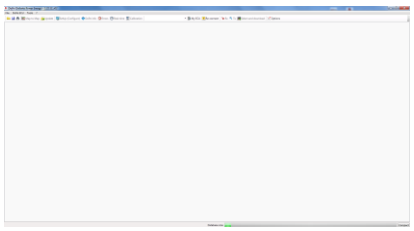
3 DoPe software

This program manages the engine configuration, and so:

- modify engine parameters, regulation and curves.
- Rx and Tx the DoPe ECU configuration
- Do TPS and DC motor potentiometer tunings.

DoPe Desktop Icon

Double click on the icon to open the DoPe software.



3.1 Select the ECU

First of all it is necessary to select your ECU to tx or Rx a configuration.

Power on the DoPe ECU and connect it to the PC with the Ethernet wire.

The "ecu finder tray" process will appear as a magnifying glass icon in the lower right corner of the screen when the ECU is powered and connected via Ethernet.

If instead of the magnifying glass icon a yellow triangle icon

is shown, right click on it and select "refresh device".

Two possibilities:

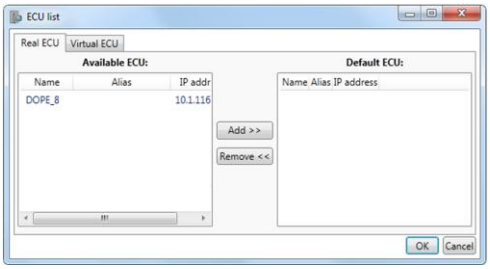
1. First time you connect the ECU to this PC.
2. The ECU is already in your “database”.

First time you connect the ECU to this PC

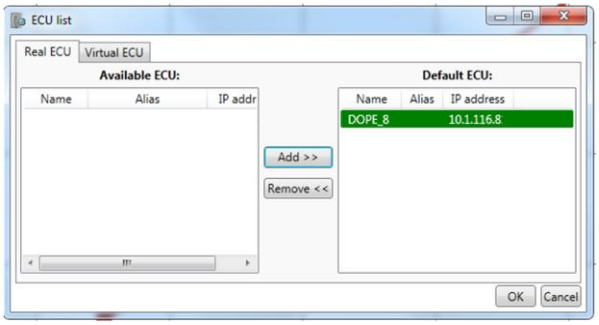
It is necessary to add the ECU to the Default list, so click on ***My ECU***

In the next window you see:

- In the left side (Available ECU) there is a list with the ECU connected to your PC.
- In the right side (Default ECU), the list show all our favorites ECU (green highline if connected)

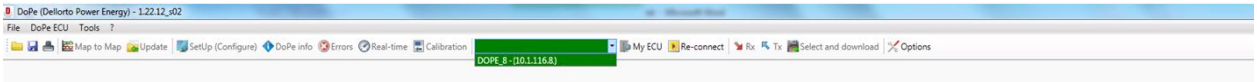


To add ECU to your favourite list, just click on the ECU (list on the right side) and then click on ***Add>>*** and then ***Ok.***



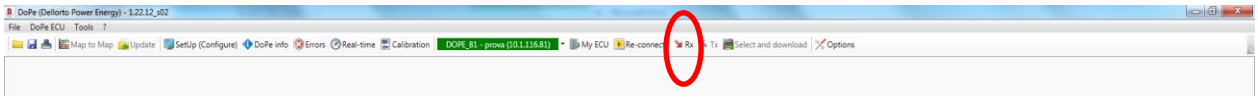
The ECU is already in our favourite list (Default ECU):

By connected the PC to an ECU thar is already in the Default ECUs list the menu becomes green to select it.



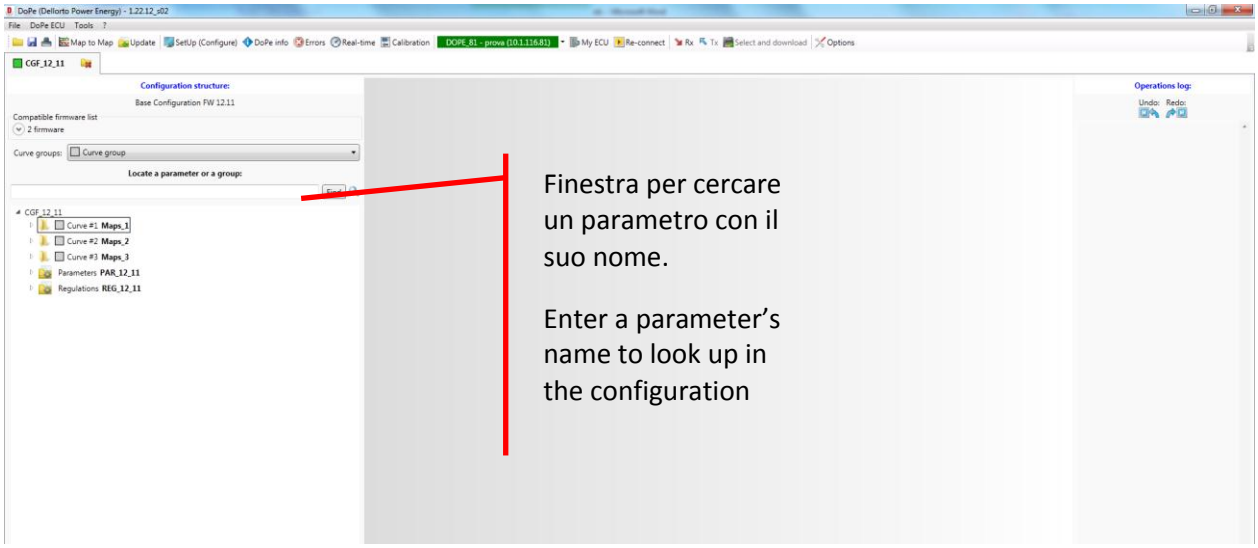
3.2 Rx the ECU

Once your ECU is selected and you see its name in the green menu, just click on the ***Rx*** option**.**



La configurazione si scarica e si mostra alla destra:

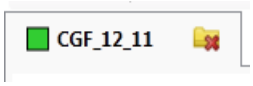
The downloaded ECU is on the right side of the screen:



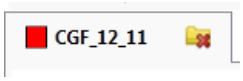
3.3 Tx the ECU

Important! To transmit a configuration to the ECU, this must be compatible with ECU firmware.

To check the compatibility: the sqaure near the name indicates:

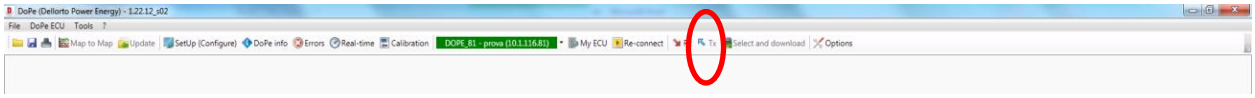


The engine is compatible with the ECU firmware version



The engine configuration is not compatible with the ECU firmware version.

Now, you can transmit the configuration just by clicking on the ***Tx*** option.



3.4 Configure the Throttle Position Sensor, Gear Position Sensor and DC Motor

***** It is highly recommended to make the tuning of the Throttle Position Gear Position (if used) and DC Motor (if used) sensors in the following situations: *****

- New ECU DoPe
- When formatting the ECU DoPe or re-programing firmware. (Check also the calibration maps value)
- When change the sensor or the device

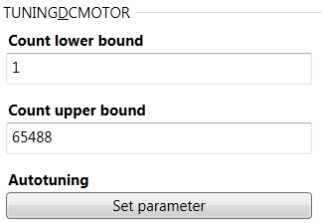
This is strictly necessary for the correct working of the system.

1. Click on the SetUP (configure) option in the DoPe menu to show the corresponding window.
2. Connect the ECU to the PC and switched it on.
3. Open MARdata software then open MAT and select the ShowRealTime option.
4. ***Gear Sensor Set up:***
    a. Look for the Gear value (in counts) of each gear.
    b. Write each gear count value in the corresponding GEAR of the GEAR_THRS parameter in the DoPe configuration.

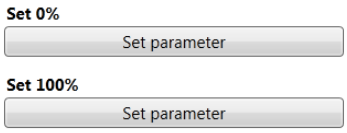
    You need the Internal channel for the Gear Sensor in the acquisition table of the DoPe ECU. So create (if not exist) an Internal channel to the analogic channel that you connect the gear position sensor (GEAR_USE_SENSOR DoPe parameter):
    - CH_A5V5 if connected to AN5V5 pin
    - CH_A5V9 if connected to AN5V9 pin

    Obviously, you just need to set the gear position sensor if used it and therefore GEAR_USE_SENSOR is set to connected to AN5V5 pin or connected to AN5V9 pin.

5. ***DCMotor set up:***
   a. In the SetUp (configure) window in DoPe software click on the corresponding Set parameter button



6. ***TPS (and TPS Aux if used) Set up:***
   In the SetUp (configure) window in DoPe software:
   a. Leave the throttle at 0% and click on the Set0% SetParameter button.
   b. Full throttle and click on the Set100% SetParameter button.

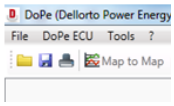


3.5 How to compare two configuration

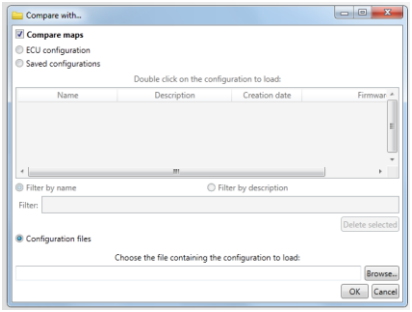
You can compare 2 DoPe configurations.

Compare the acttive configuration with a configuration saved in your PC.

Click on Map to Map in the menu.



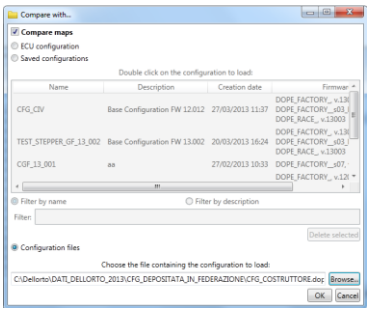
Next window will be open:



Compare maps: select this option to compare also the values of the map curves.

Select ***Configuration files*** option to choose the configuration to compare in your pc.

When you already fill in all, click on Ok button.

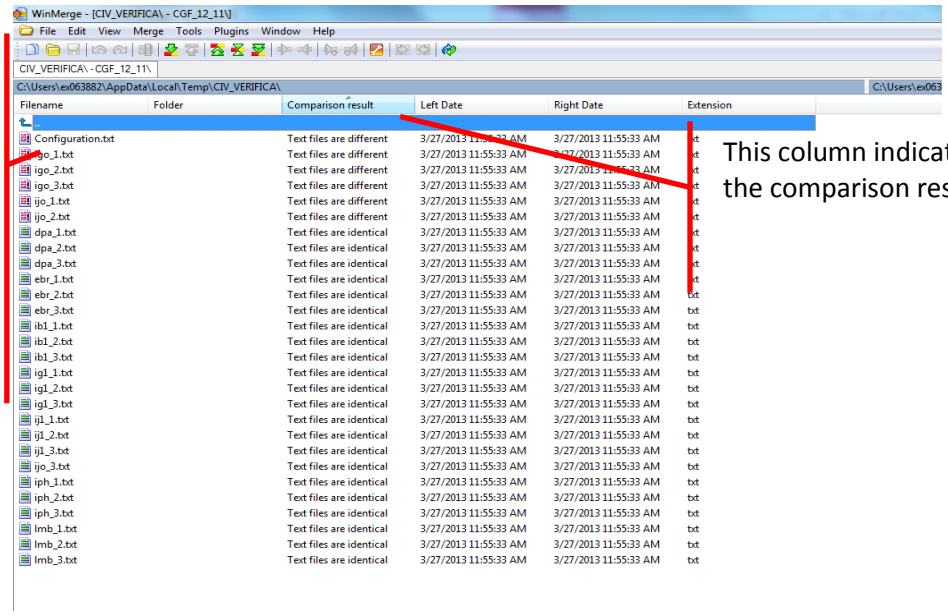


WinMerge software will be opened. This program compares the selected configurations.

This list means::

A Configuration file to visualize every parameter and regulations

The other files are for the maps curves.



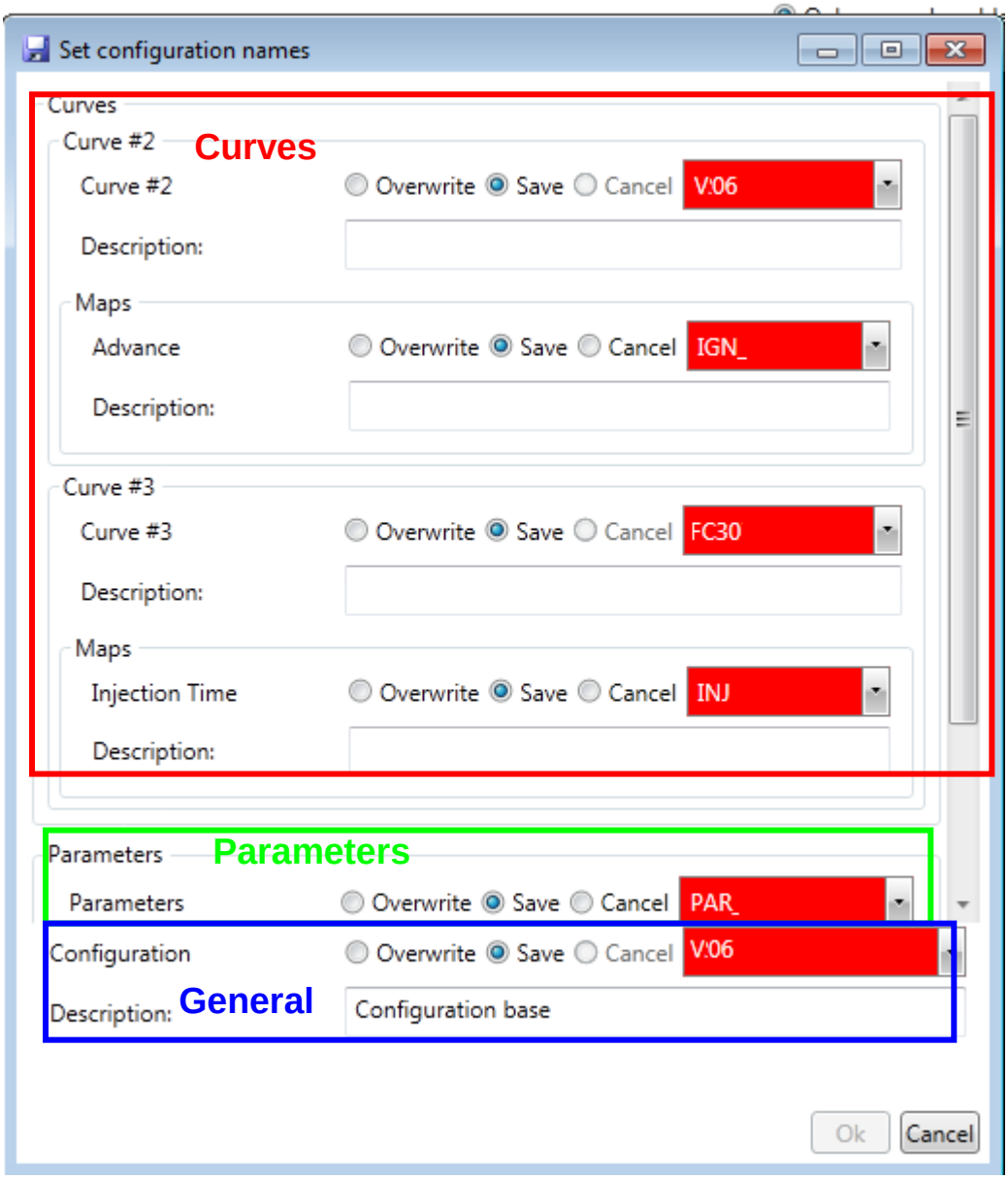
3.6 Saving the new configuration

The next two options are available by clicking the “File” option in the main Toolbar and then “Save”:

- Save Database Ctrl+S
- Export file…

1. Save configuration to database

Before transmitting a configuration to the ECU it is recommended to save this configuration to database. Click on the “Save configuration to the database” option or use the shortcut <Ctrl+S> or the icon. A similar window appears:



"Set configuration names" window

All modifications are divided into groups like the main configuration structure is organized.

- *Curves*
- *Parameters* → Tuning adjustments.
- *General*

A. Curves

Within this section there is a sub-section for each group of curves that has been modified (curve #1, curve #2,etc…).

Within the subsection "*Curve#*" the subsection *Maps* indicates which maps have been modified.

B. Parameters

This section appears if you have modified one or more parameters in the parameters option.

C. General

This section refers the whole configuration.

How to save the configuration

The user should confirm the modifications completed during the session.

There are three radio button options:

1. ***Overwrite:*** save with the same name but losing the previous one.
2. ***Save:*** Is like the "save as…" option. With a new name. This option saves modifications or new configuration in the database, without losing the previous ones.
3. ***Cancel:*** this option cancels only the selected section. For example, in the previous figure, by checking cancel in the *Parameters* section only these modifications will not be saved.

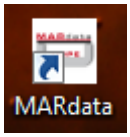
2. Export configuration to file

This option saves the configuration in a folder. Click the "File" option in the main Toolbar and then "Save". Click on the "Export configuration to file option. This option also allows the user to export the configuration from one computer by saving a single file in order to use it in another computer.

4 MARdata software

MARdata is the software developed by Dellorto to download the data from the ECU, administer the acquisition files and analyze it. Also allows administering, creating, modifying, transmitting and receiving the acquisition table.

Desktop Icon



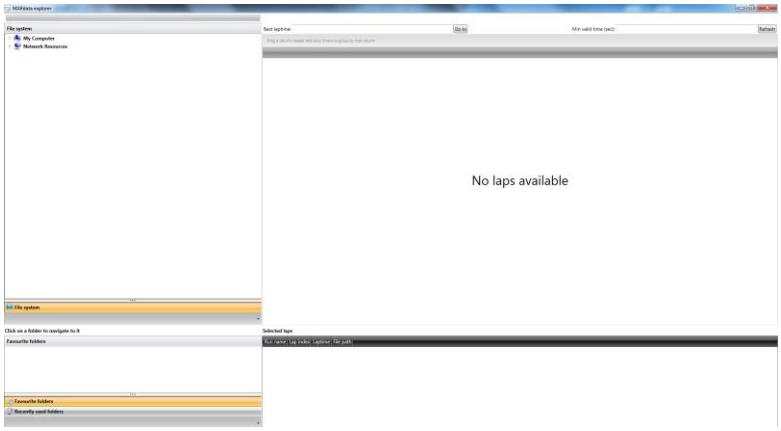
The principal window will be open by clicking twice on this icon.

MARdata: to administer the acquisition files and analyze them. (.dop extension files).

MAT: to administer the acquisition table (.mat files)

Click twice on the MARdata option.

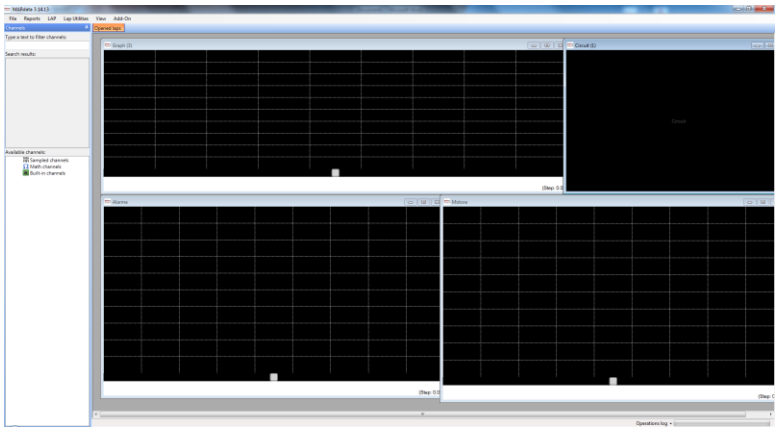
MARdata and MARdata explorer will open. The MARdata explorer window administers the acquisition files already saved in your pc.



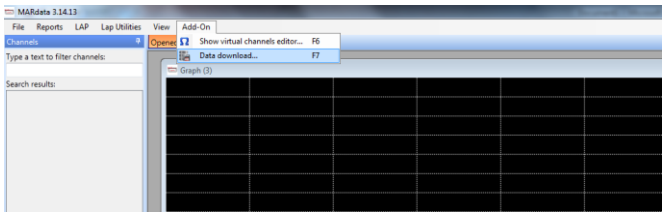
But first of all we explain how to download the acquisition files from the ECU, so you can close this window.

4.1 Download the acquisition file from the ECU DoPe.

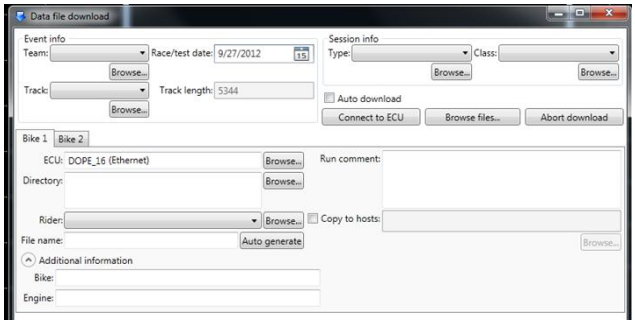
Next figure shows the main window, where analyze the acquisition files.



To open the Data Download windoy click on the menu option ***Add-On*** then ***DataDownload.*** (Shortcut ***F7***)



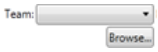
Next window is the ***Data file download*** window



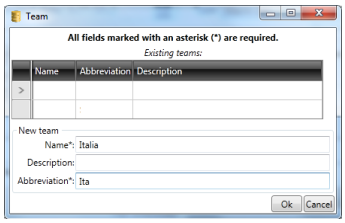
4.2 How to fill in all the fields of the Data Download window:

***IMPORTANT*: the characters which are admitted are the letters from 'A' to 'Z' and from 'a' to 'z' of the standard Latin alphabet, all numbers and the hyphen (-), point (.) and comma (,) symbols.**

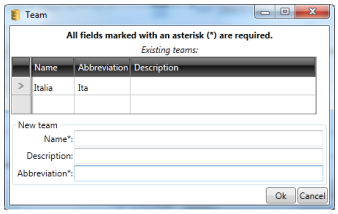
First group to fill in: Event info and Session info:

TEAM: 

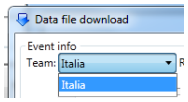
1. If it is the first time you download a file from a team: Click on the ***Browse…*** option to enter the information of this new team. Fill in the ***Name*** and an ***Abbreviation***, and then click on ***Ok*** button. * options are mandatory.



Next time you click on ***Browse…*** all the team's name that you already wrote will be listed.

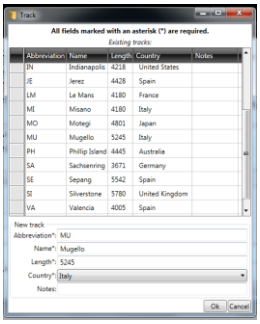


2. If the team's information is already saved: just click on the ***Team*** option and choose the desired Team name.

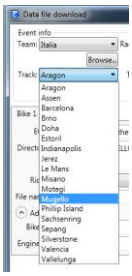


TRACK:

1. First time to run in this track: Click on the ***Browse…*** option to enter the information of this new track. Fill in the ***Name***, an ***Abbreviation***, the ***Length*** of the track and the ***Country*** then click on the ***Ok*** button.
   * options are mandatory.
   Important!!! By beginning the acquisition file's name with the Abbreviation the corresponding track will open in the track window graph (if exist in the layout) when you open the acquisition file. For example Abbreviation = **MU** acquisition file = **MU**14ABC001A



2. If the track's information is already saved: just click on the ***Track*** option and choose the desired track name.

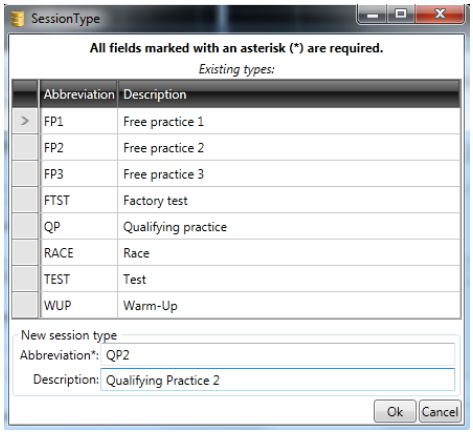


SESSION INFO e CLASS:

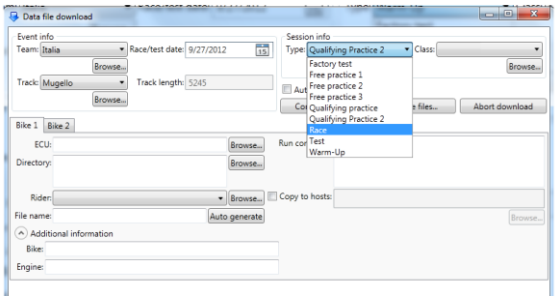
In the same manner as previous fields, fill in the ***Session Info*** and ***Class*** options.

Session Info

Write new session.

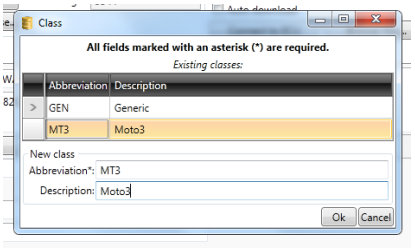


Select a session

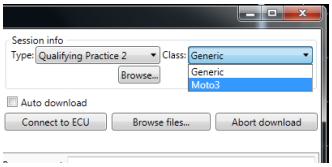


Class

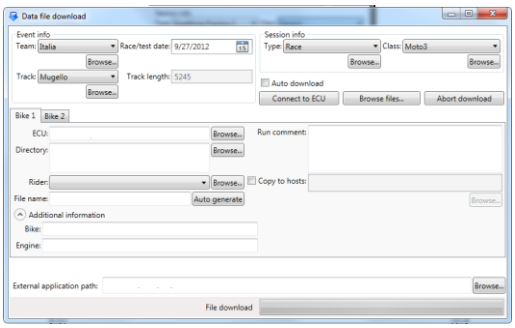
Write new class.



Select a class

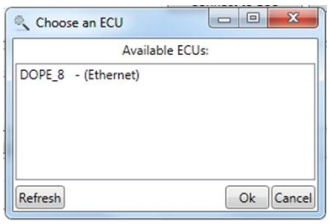


Now fill in the **Bike** tab:

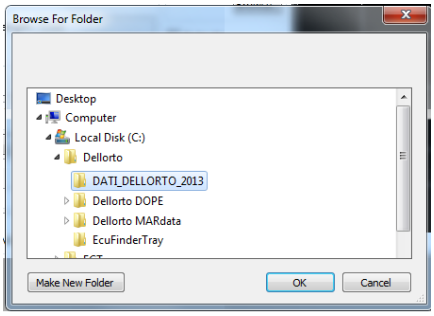


ECU: Click on the Browse button and select the ECU name to download the data. (The ECU must be switched on and connect to the PC).

Click on the desired ECU and then Ok.

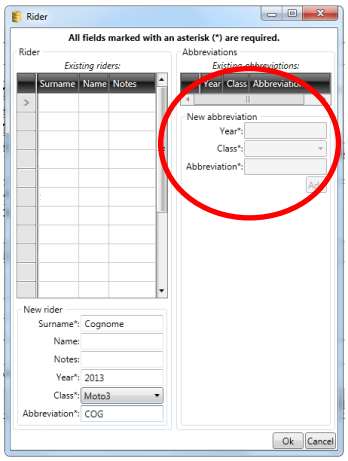


DIRECTORY: Click on the Brwose option to select the folder to save the acquisition file.



RIDER:

1. If it is the first time that you download the acquisition file of this rider: Click on the ***Browse…*** option to enter the information of this new rider. Fill in the ***Surname***, an ***Abbreviation***, the ***Year*** and the Class then click on the ***Ok*** button.
   * options are mandatory.



2. If the rider's information is already saved: just click on the ***Rider*** option and choose the desired rider.

FILE NAME: Write the name of the acquisition file.

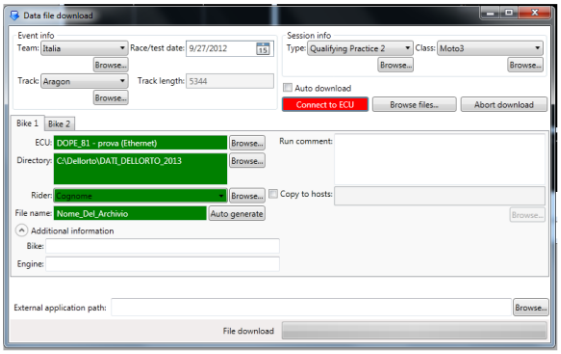
Important!!! By beginning the acquisition file's name with the Abbreviation the corresponding track will open in the track window graph (if exist in the layout) when you open the acquisition file. For example Abbreviation = **MU** acquisition file = **MU**14ABC001A

All of these fields (except the ECU option) can be filled in also if the ECU is disconnected to the PC.

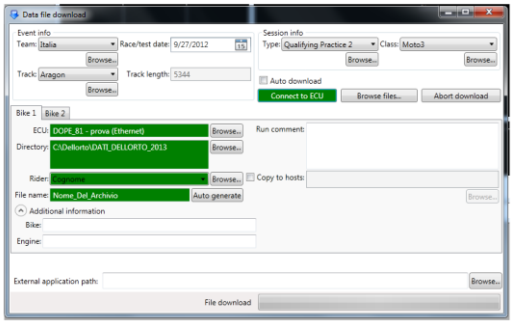
Download the acquisition file

When all the fields are filled in, click on ***Connect to ECU***.

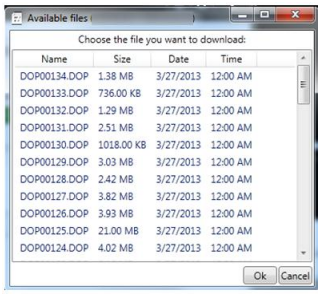
If all the fields are correctly filled, they become green and the ***Connect to ECU*** will be red until you connect the ECU to the computer.



When you connected the chosen ECU to the PC the ***Connect to ECU*** button become green and you can click on the ***Browse files…*** option to select the file to download.



Click on ***Browse Files…*** the following window will list all the acquisition files saved in the ECU. The first file in the list is the last saved Run.

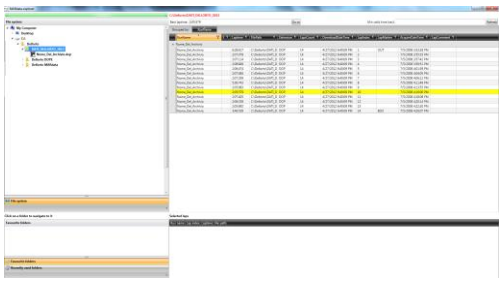


Select the file and then click on ***Ok***. When the download process finish, the acquisition file will be opened in the MARdata window to analyze it:

4.3 How to open an acquisition file that we have already saved in the computer

In the MARdata window (previous figure)press ***F3*** to open the ***MARdata explorer w***indow.

In the left part of the window is the explorer with all the folder of your computer. By selecting the folder all the .dop files (if exist) will list in the right part of the window.

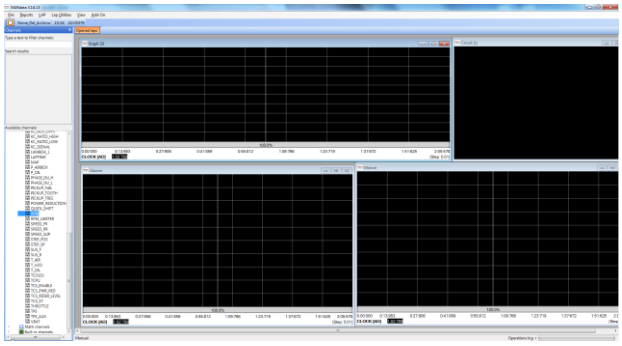


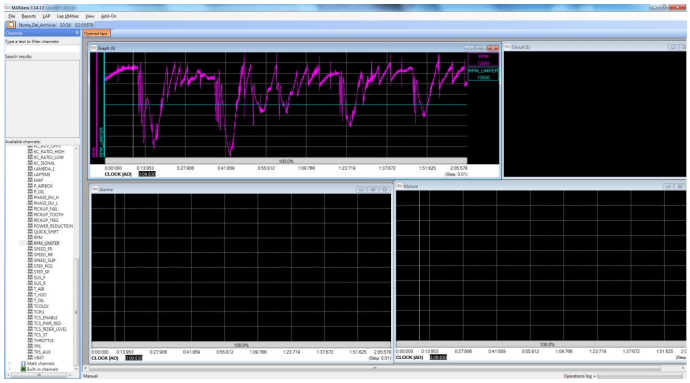
Click twice on a file to open it in the MARdata window to analyze it.

4.4 How to add channesl to the graph.

In the left part of the MARdata window there is a list with all the channels: Channels option.

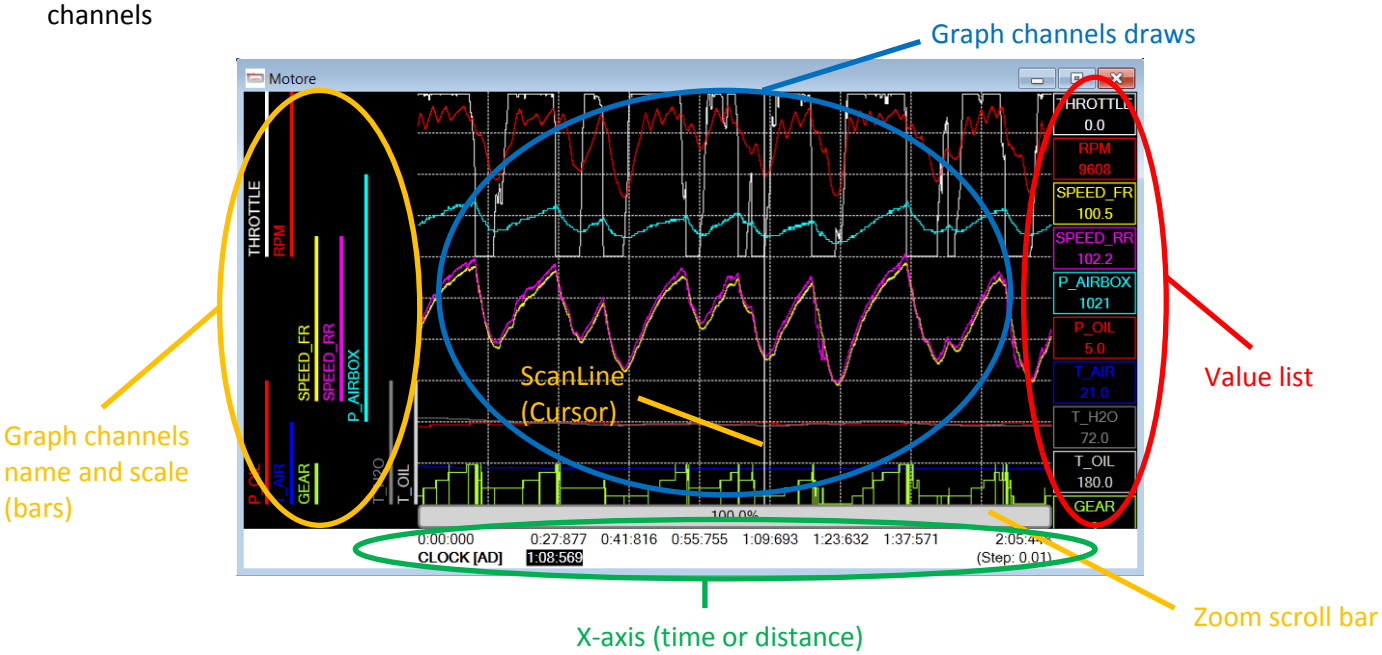
First of all select the graph to add the channel, then select the channel and click twice on it.





4.5 Graph

Next example is used to explain the principal properties of the graph files and the properties of the added channels



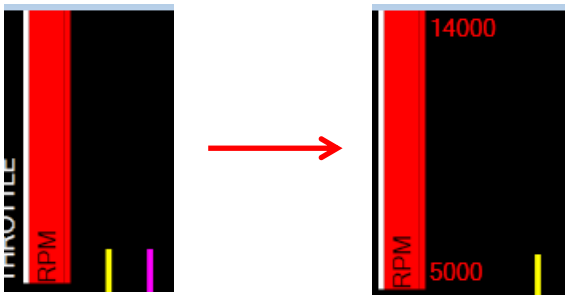
- ***ScanLine*** is the main graph tool. This tool allows to use the standard cursor in the graph. Place the Cursor in the graph and see the channel values in the value list at the right part of the screen. To change the ScanLine Cursor position, just click the mouse left button and move the mouse cursor holding the button down. In order to select this tool click on the "Lap Utilities" item in the main menu and then select the Scan line option; the shortcut <Ctrl+Shift+L> can be used too.
- The portion of visible data can be changed using the ***Zoom*** tool: to zoom and see a portion of logged data, click the mouse right button, move the mouse cursor holding the button down to select a new data region, release the button and then click Zoom. Zoom operation can also be performed from the option Zoom under "Lap Utilities" item in the main menu.

  Refer to *ScanLine tools* section and *Zoom tools* section for more information about ScanLine and Zoom.

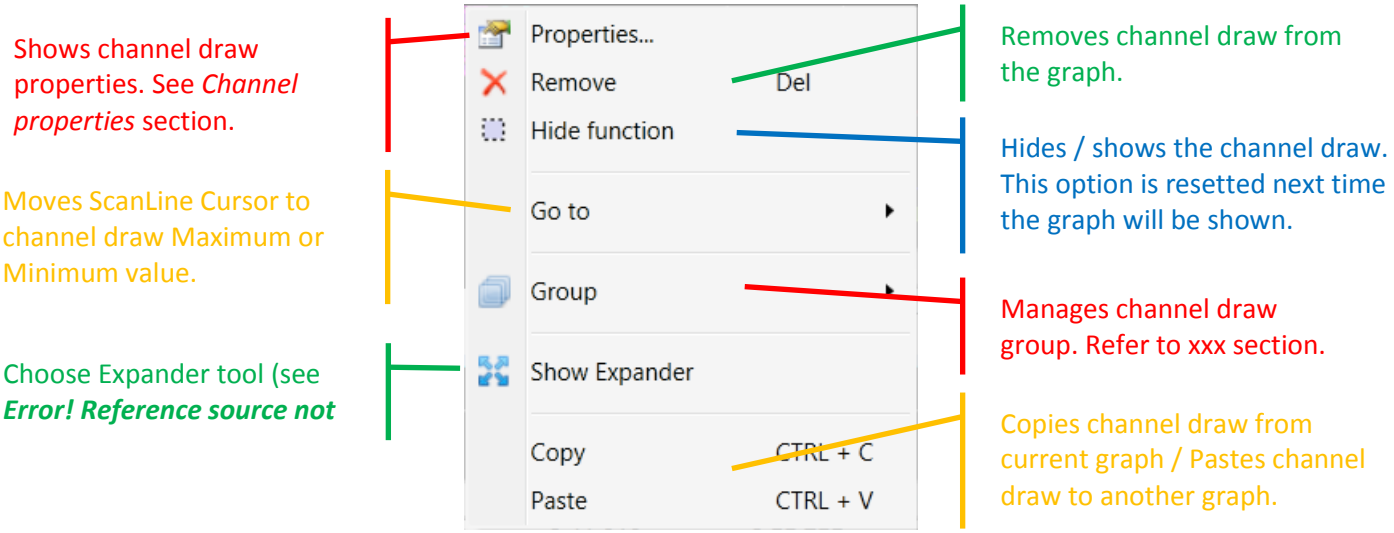
- ***Zoom scroll bar*** indicates the zoom percentage of the graph. Click and hold on the bar and with the mouse scroll left and right through the lap.
- ***X-axis***: the graph may be represented as a function of time or distance. Click on the "View" main menu item, then X axis option and choose the x-axis representation.

4.6 Channels name and scale (bars)

Each graph channel has a *Channel bar* (on the left of the graph) showing channel name, color and the portion of the graph the channel uses. Click with the left button of the mouse to select one (and highlight) one or more graph channels. Middle button (wheel) clicking with the mouse on a bar will toggle channel bounds (maximum and minimum values) visibility:



Some other operations can be performed using the *Channel bar* context menu:



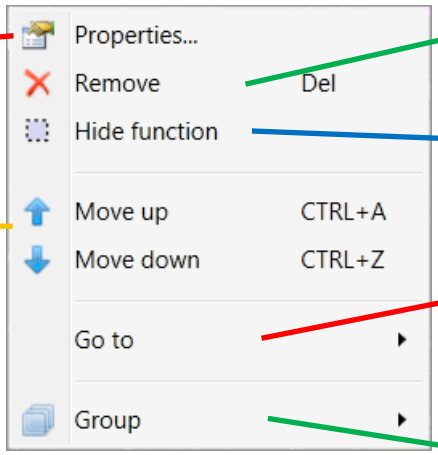
4.7 Channels value list

Each graph channel has a *Channel value* (on the right of the graph) associated to it, showing the value the channel has in the current ScanLine Cursor position; cursors values can be shown too (see *Graph cursors* section). Click with the left button of the mouse to select one (and highlight) one or more graph channels.

Some other operations can be performed using the *Channels value list* context menu:

Shows channel draw properties. See *Channel properties* section.

Moves channel draw to bring it to the Foreground (Move DOWN) or to the Background (Move UP), with respect to other channels shown within the graph.



Removes channel draw from the graph.

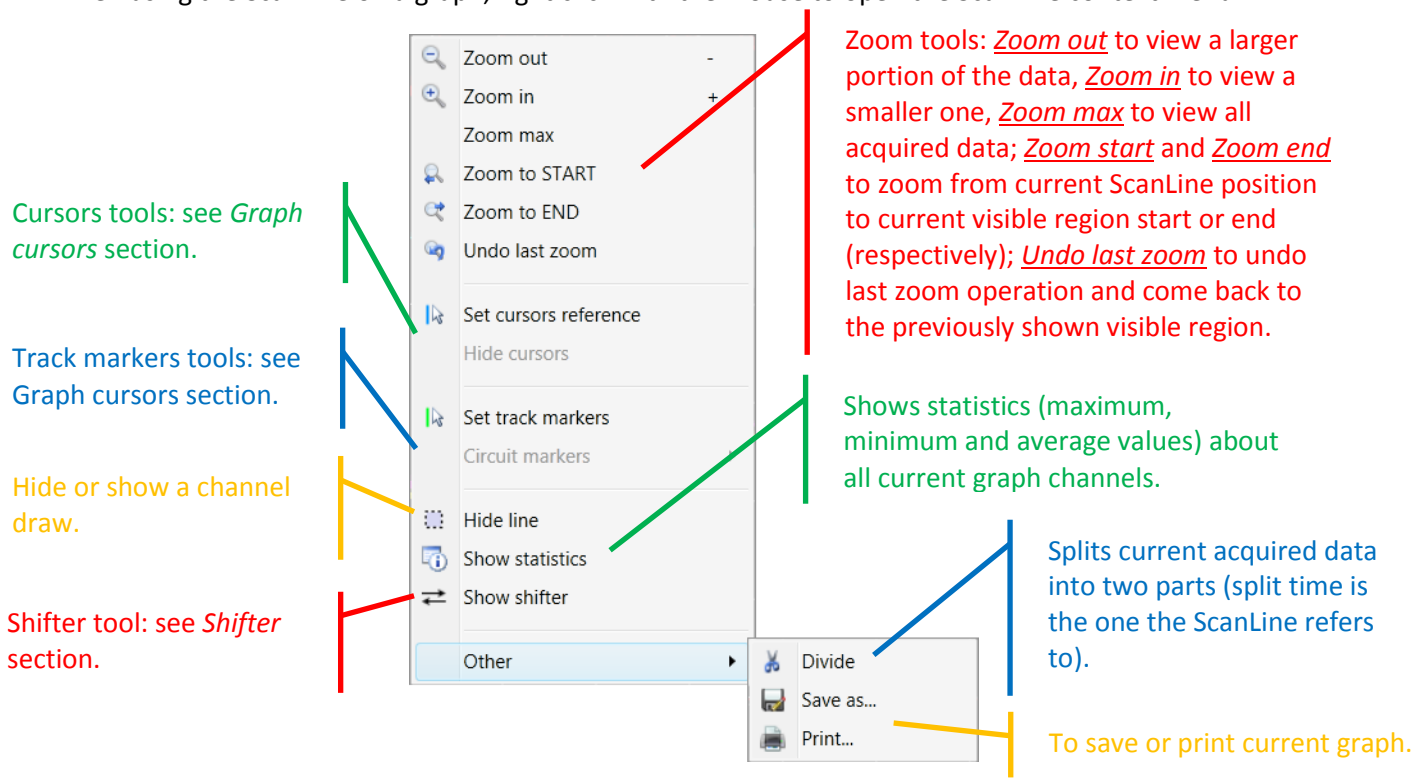
Hides / shows the channel draw. This option is resetted next time the graph will be shown.

Moves ScanLine Cursor to channel draw Maximum or Minimum value.

Manages channel draw group. Refer to xxx section.

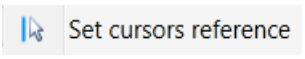
4.8 ScanLine tools

When using the ScanLine on a graph, right click with the mouse to open the ScanLine context menu.



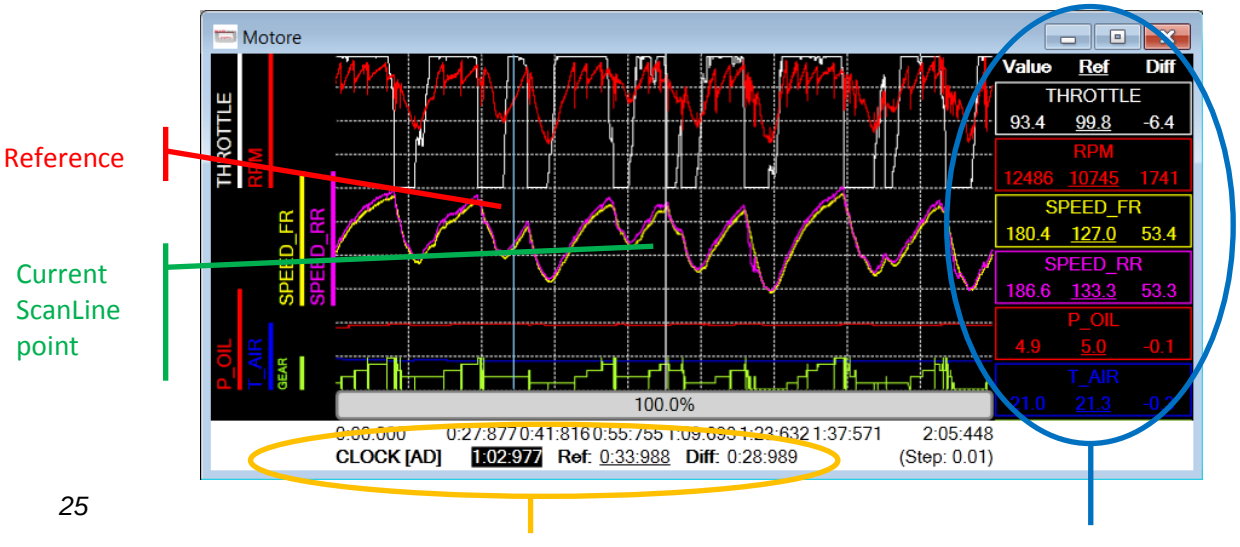
4.9 Graph cursors

Graph cursors are used to check differences on a graph channels values between two points: the current one and the so-called reference one.

To choose a reference time/distance point, move the ScanLine to that point, right click and press 

. A blue vertical line appears when the ScanLine is; to change the reference point, move to another time/distance and repeat the operations.

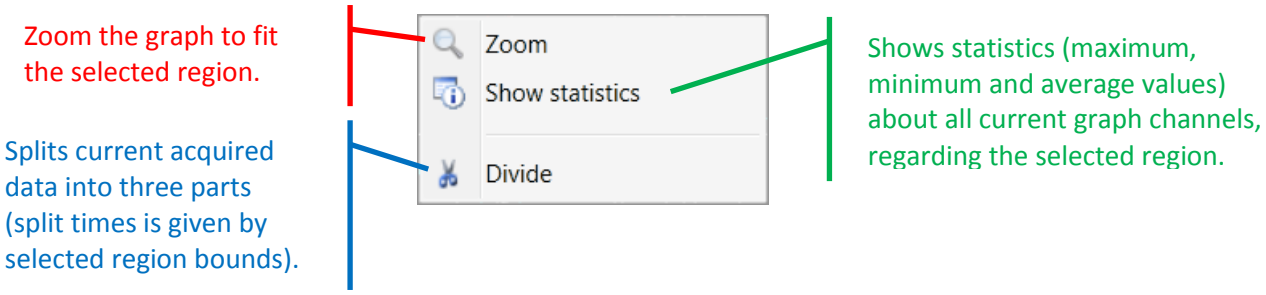
When a reference is set, differences between the reference and current ScanLine point are shown:



To hide cursors, right click and press Hide cursors .

4.10 Zoom tools

When using the ScanLine on a graph, a new region can be selected right clicking and moving the mouse while holding the right button down; when the button will be released, zoom context menu will appear.



Another way to change data visible region is using the *Scroll bar*; when zoom factor is less than 100%, clicking with mouse left button on the scroll bar and moving the mouse (holding the button down) will cause the zoomed region to change; double click with mouse left button on the scroll bar to restore the zoom factor to 100% (i.e.: the same of choosing Zoom max from ScanLine context menu).

Zoomed region can also be changed using the *mouse wheel*: scrolling the wheel will cause the zoom region to mouse leftward or rightward, while scrolling the whell *holding the CTRL key down* will cause the zoom factor to change.

4.11 Channel properties

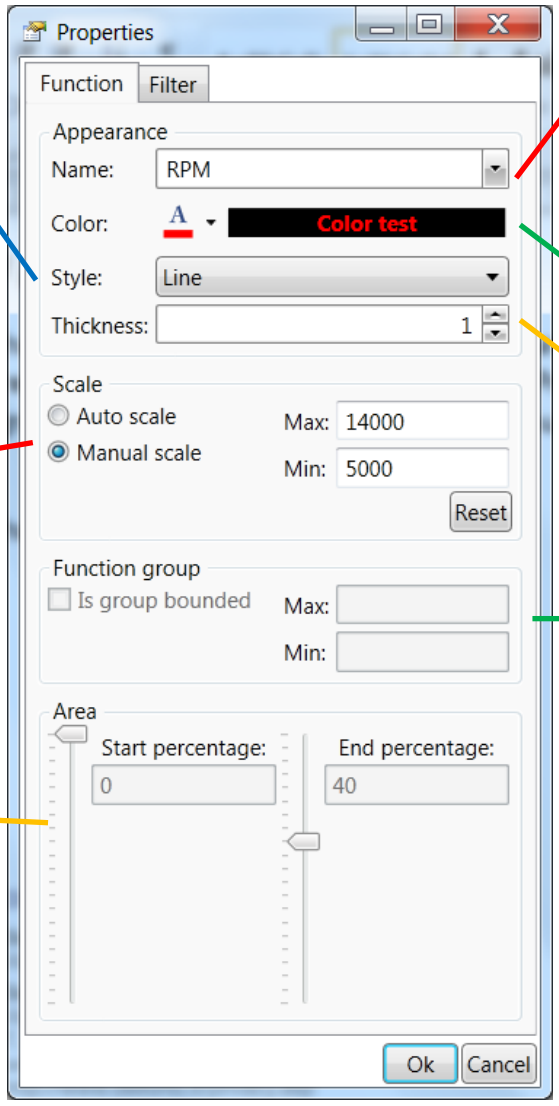
To view a drawn channel properties, double click on the channel name on the right of the graph (where channels values are shown), or right click on it and choose *Properties…* (the same is valid even for the bar on the left of the graph, where channel name, color and bounds are shown); the Properties window will appear.

Property window has two sections: *Function* and *Filter*.

Channel draw style. Available styles are *Line*, *Step* and *Dot*. Refers to section ??? for more information. Remember that *Dot* style is reccomended only if used with high zoom factors.

The scale used to represent channel values. *Auto scale* means that the draw adapt its maximum and minimum shown values to fit the whole available area, while in *Manual scale* draw maximum and minimum shown values are user-defined.

The area of the graph used to represent current channel (when in MANUAL mode). O% means the top of the graph, while 100% mens the bottom of the graph.

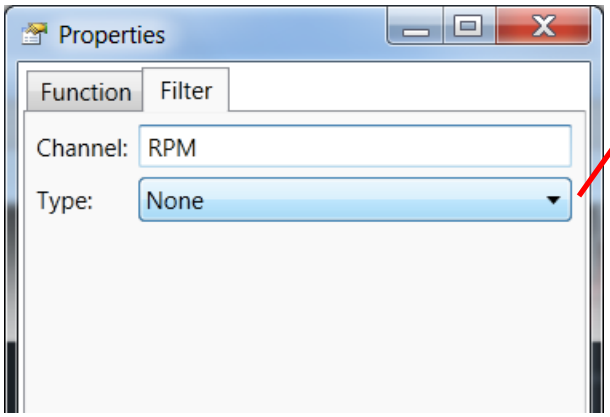


The name of the channel. Change this name to replace current channel with another one. All channel properties will be kept.

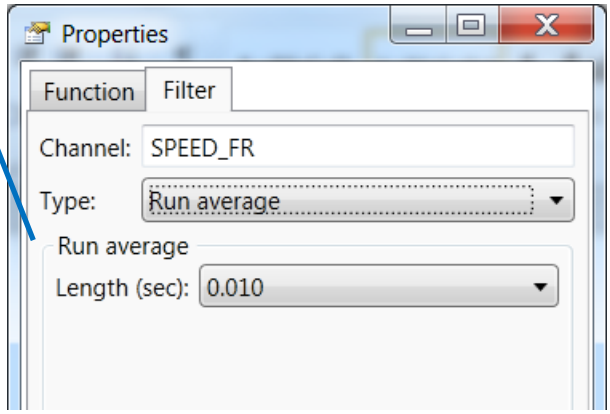
Channel draw color.

Channel draw lines (or dots) thickness. Setting this value to 0 (zero) will cause the draw to disappear.

When two or more channels (functions) are grouped (see section ???), their group maximum and minimum shown values are auto determined. Enable this section to manually specify channels group maximum and minimum values.



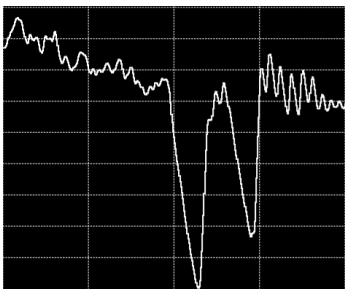
Choose *None* to disable channel data filtering. Choose *Run average* to filter channel data using a run average filter (see below).



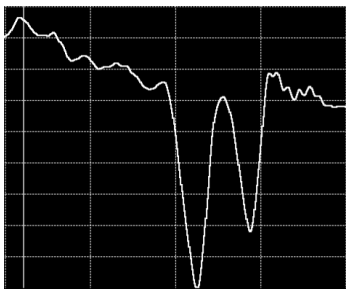
Run average filter shows the arithmetic mean of a sample. All values are summed and divided by the number of samples; this number of samples depends on the length of the filter the user choose: the higher the length value, the more the channel will be filtered.

Filtering example:

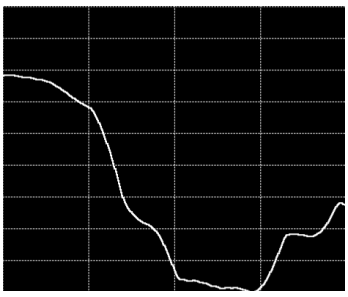
No filter



0.1 sec filter applied



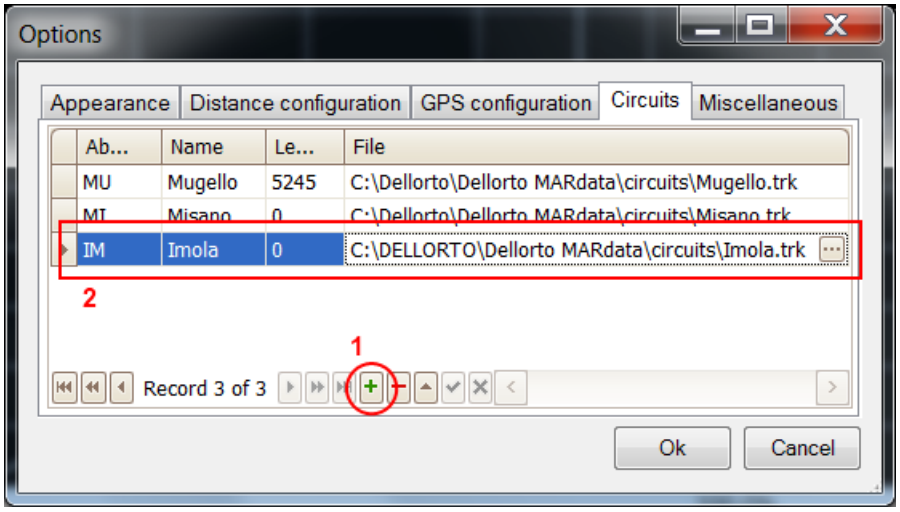
2 sec filter applied



4.12 How to move track finish line.

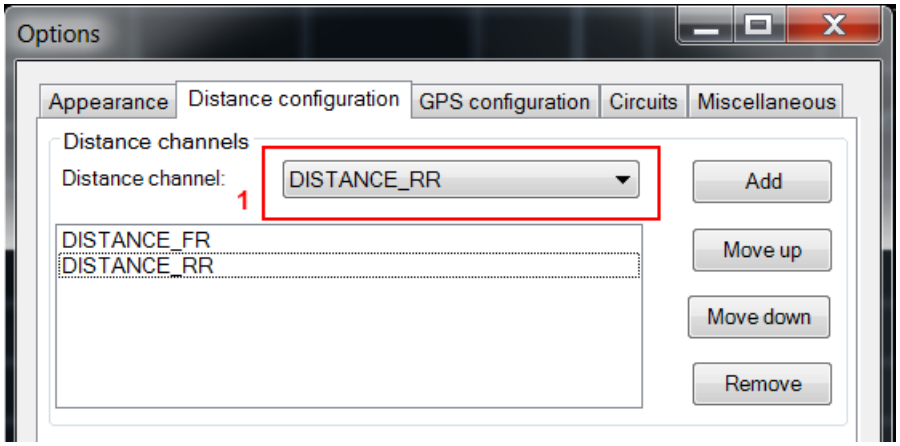
In order to move a track finish line in a Circuit Report within MARdata, the user:

1. must load some acquired data related to that track
2. must have the track file (*.trk) related to that track
3. must have done an association between the track file and the abbreviation of the track name; such an association (that is necessary to automatically load the correct track within the track Report, depending on the acquired data currently shown in MARdata) can be created clicking on File/Options…, then moving to the Circuits tab:

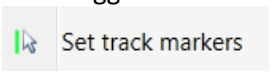


To add a new association, just click on the green plus button (1), a new empty row will be created. Just type a two letters abbreviation (this abbreviation must match the first two letters of the data file name, so MARdata can automatically load the appropriate track file within the circuit Report), the name of the track, the length of the track (optional) and choose the trk track file that contains the track map. To edit an existing association, just change a row values.

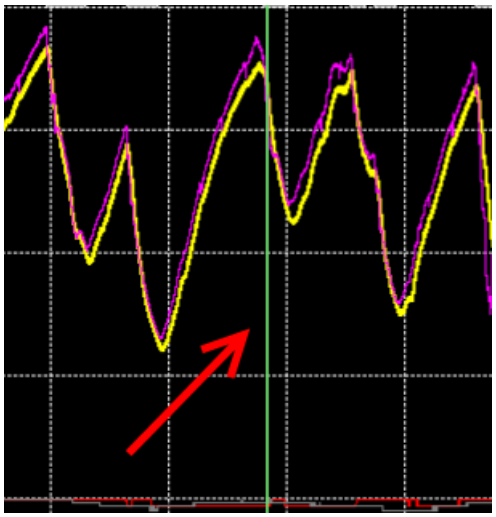
4. must have chosen a channel that represents the distance the bike went through (for example: DISTANCE_FR); to select it, click on File/Options…, then move to the Distance configuration tab:

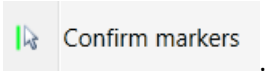


Choose the channel from the list (1), then click Add; more than one channel can be specified: the higher the position, the higher the priority (in the case above: if DISTANCE_FR is found it will be used, otherwise the program will look for DISTANCE_RR, and so on).

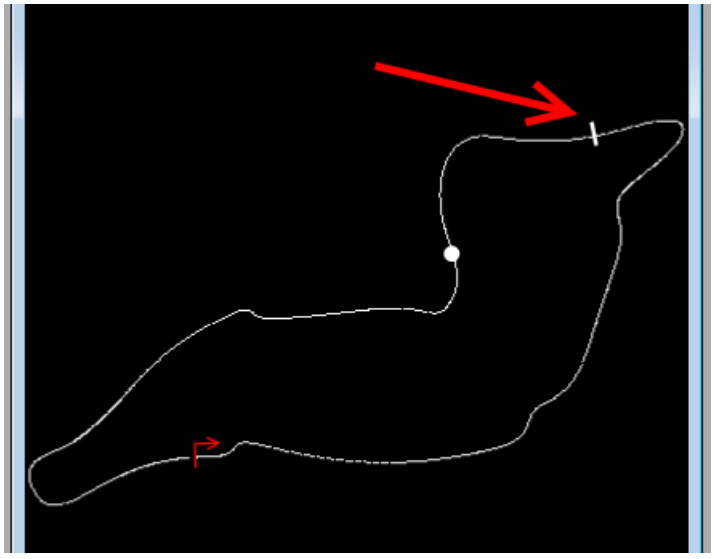
Once logged data is loaded and track is shown on circuit Report, right click on a Report window and choose 

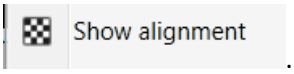
; the cursor will change, showing an arrow with a ± symbol. Choose a point on the Report and left-click holding the CTRL key pressed: a green line will appear on the chosen point



Now, right click again and choose 

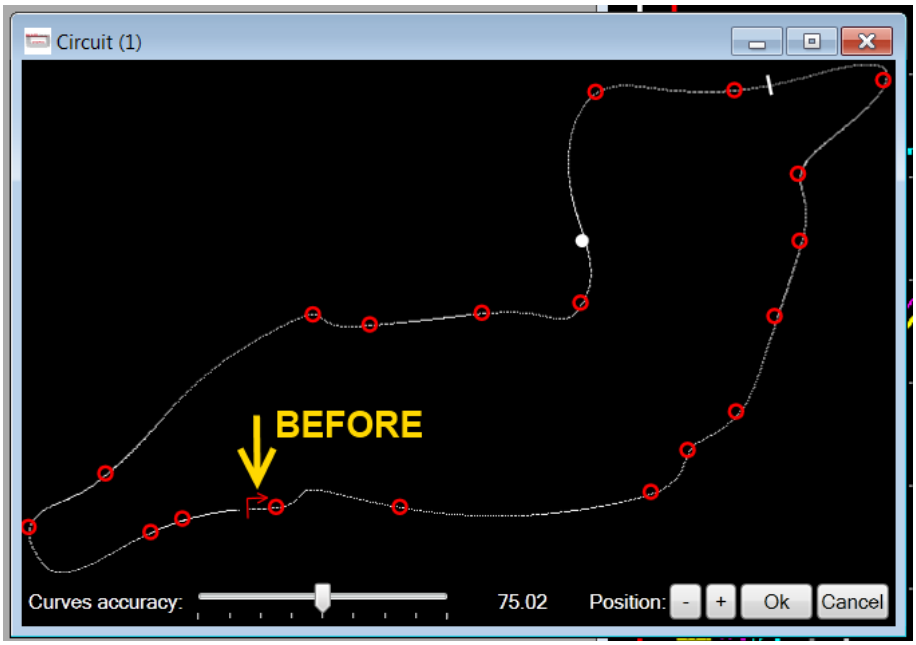
. The cursor will become an arrow again, and a line will be shown on the track in a point that corresponds to the one the you chose on the Report

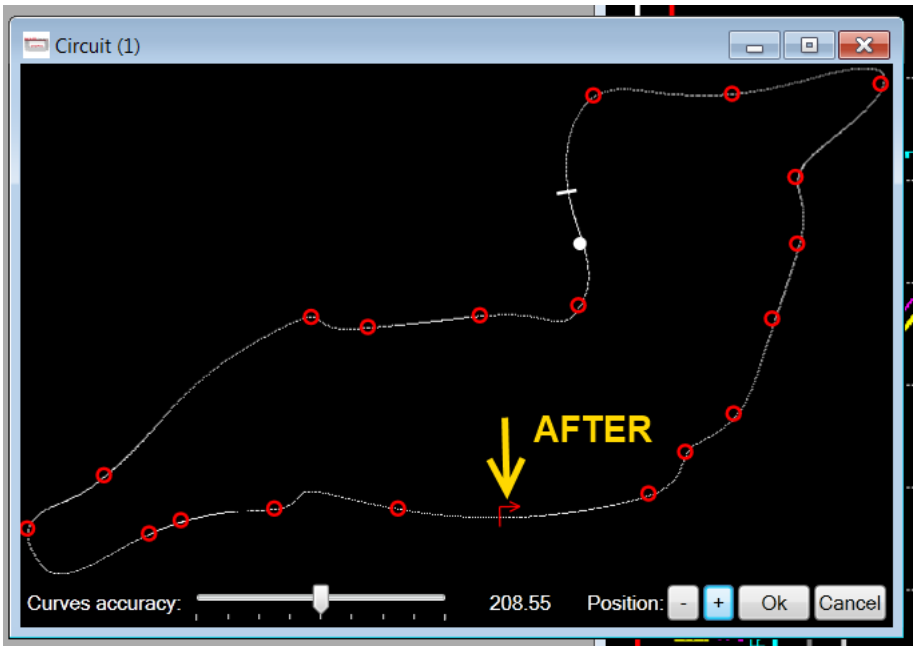


After doing so, right click on the circuit Report and click on 

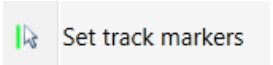
. The alignment controls will be shown below track map; use the buttons near Position (

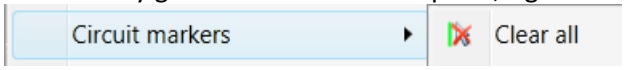
) to move the finish line forward or backward

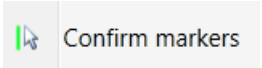




Once the finish line is in the correct (new) position, click on Ok. To save permanently the change, choose Reports / Circuit / Save circuit file as… on the main menu, then choose the file where to save the new trk data.

To clear any green marker on the reports, right click on a report and choose 

, then 

(answer “Yes” when prompted for confirmation) and, finally, click 

.

If you want the new track to become the new default one for the current circuit, remember to edit the circuit association, as explained above.