Emeraude (CHL) Tutorial #1 – Conventional PL & Emeraude Introduction

1. Introduction

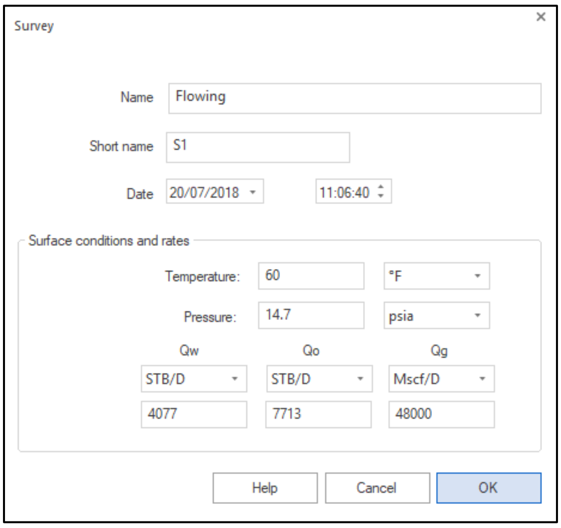
This session illustrates the basic software features and the interpretation workflow offered by Emeraude.

The data comes from a nearly vertical well, logged with a Fullbore Spinner, Water hold-up probes, Gas Hold-up probes, Nuclear density, Pressure, Temperature and Caliper. The flow is 3 phases from a flowing survey. The flowing passes, after a data quality check, will be processed to obtain the flow profile and zonal contributions, using different calculation methods. The hold-up measurements correspond to the arithmetic averages of the optical and electrical probes, and they will be used directly, not going through the MPT (Multi Probe Tool) processing. The emphasis is on the software mechanics rather than interpretation methodology.

The necessary data files are located in the 'Examples' folder under the 'Emr540' directory (by default in C:\Program files (x86)\KAPPA).

2. Creating a New Document

Click on the 'New', icon to create a document, from the File tab or the application toolbar. In the Information tab, ensure that the type of interpretation is set to PL. You don't need to fill the other information. Go to the Document Units, and check that the unit system is 'Oil field'. Click on Next > to proceed.
The Survey information dialog will pop up, prompting us to provide some details about the new survey. In name, write 'Flowing'. Enter the surface rates Qw= 4077 stb/d, Qo=7713 stb/d, Qg= 48000 Mscf/d.



3. Load Well Data

The general well data will be entered in 2 ways: manually and from logs.

Click on the ‘Details’, icon from the Well tool strip. In the ‘Internal Diameter’ tab define a constant ID as [0 ft. – 6.184’’].
In the Perfos tab, define the following intervals [14900 – 14940ft], [15008 – 15080ft] and [15096 – 15125 ft.]
In the Markers A tab, we will define the following depths and marker’s name: [14896 – Top Sand 1], [15007 – Top Sand 2], [15093 – Top Sand 3]. Check the ☑ box, next to the marker’s name column.

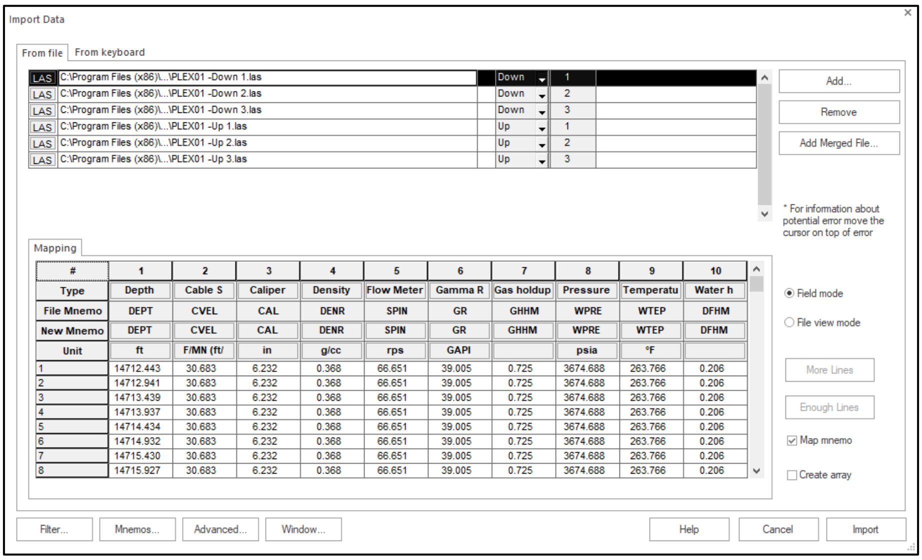
We will proceed loading logs, using the Load data option. Find the ‘CHLEX01 – General well data.las’ file, which contains TVD vs MD, and Gamma Ray information. Click on Import.
A specific track called ‘Z’ (for ‘Zones display’) is created to show, by now, only the perforations. The Gamma Ray and ID channels are also automatically created, and displayed in the corresponding tracks.

4. Load PL data

Emeraude can load data in DLIS, LIS, LAS, or ASCII format. The load option automatically recognizes the file format and decodes it as appropriate. When nothing in the file is recognized, the inconsistency is pointed out and the user is prompted to correct it. In this session the survey data consists of 6 passes, 3 up and 3 down. They are physically stored in Log ASCII Standard (LAS) files and the file names indicate their type and index: CHLEX01 – Down1.las, CHLEX01 – Down2.las, CHLEX01 – Down3.las, CHLEX01 – Up1.las, CHLEX01 – Up2.las, CHLEX01 – Up3.las.

Click on the ‘Load’, icon from the Survey tool strip and Add... . Select the 6 files PLEX01*.las. You can select all files at once with the ‘Shift’ key pressed or one after the other with the ‘Ctrl’ key pressed. You can also click on ‘Add...’ every time a new file is selected. Be sure to enter up, down and pass numbers as appropriate.

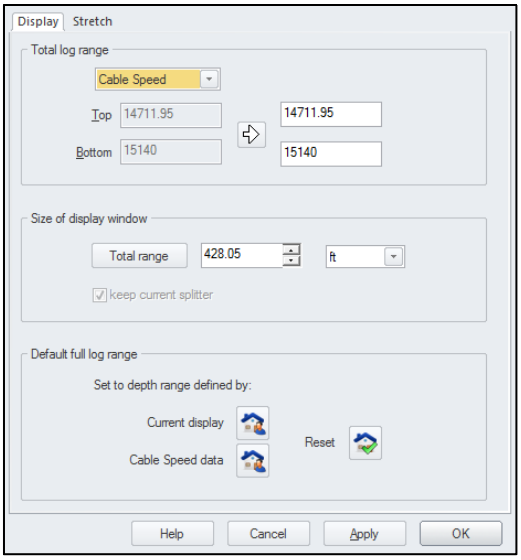
The Load dialog should look as shown below:



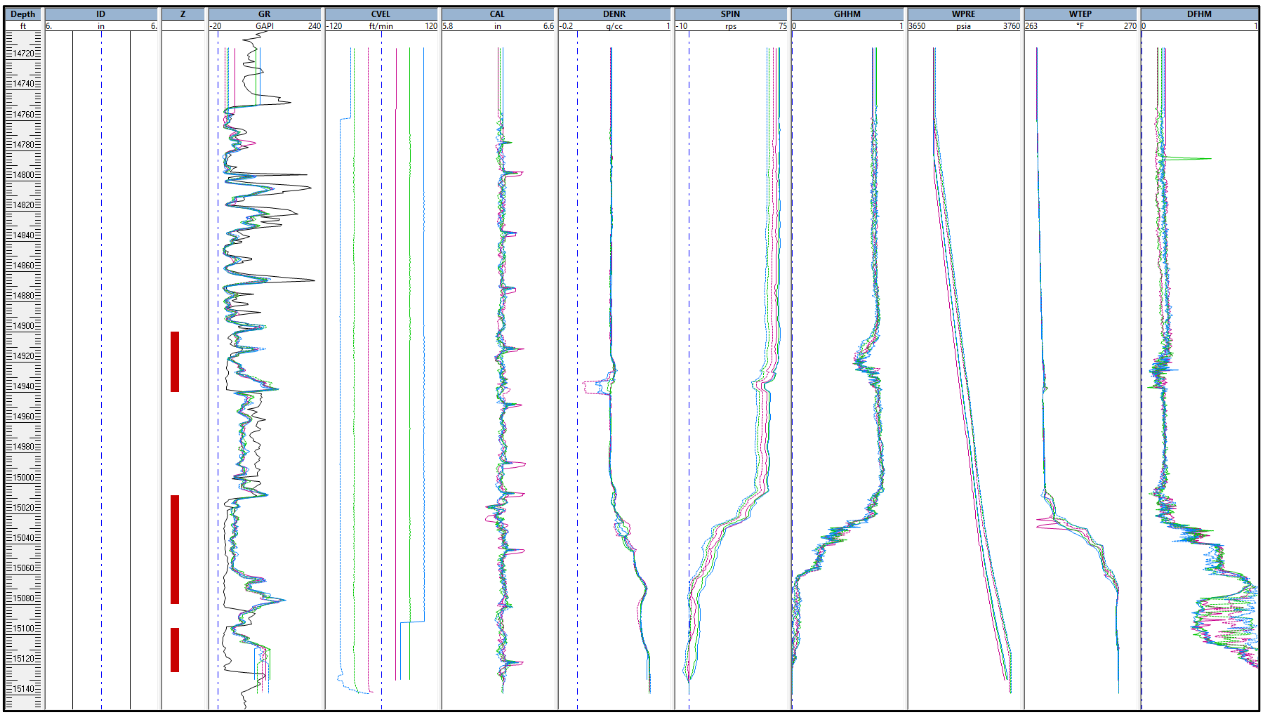
Click on ‘Import’ to execute the load.

5. Manipulating Plots

After loading the data, it's clear that the depth range is not adequate. Scale options can be accessed from the scale toolbar. Click on the 'Set depth limits', , option. We can use the cable speed to define the Total log range. Click on , and then set the size of the display window to this range, by clicking on Total range. Clicking on Apply, it can be seen that the depth range is now correct. Finally, to set this range as the new default, click on .



The tracks should look like:

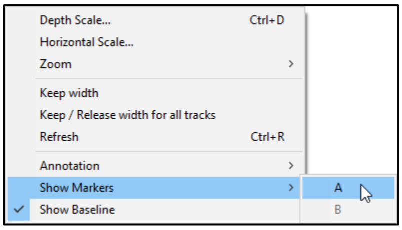


If the plots are not occupying the whole work space, use 'Tile the plots' to resize the tracks.

Clicking the right mouse button with the cursor in any plot accesses a pop-up menu. In the 'Horizontal Scale' dialog the horizontal range of the particular plot can be entered directly. When the cursor is moved onto the plots, it changes to a crosshair over the full plot area. The position (depth, value) of the cursor is shown in the message bar at the bottom of the window.

Emeraude automatically creates a plot for each channel type and each mnemonic found in the up and down passes. All channels in a particular pass are plotted with a unique aspect corresponding to the index of the pass in the type (red=1, green=2, etc) defined by the Display Settings.

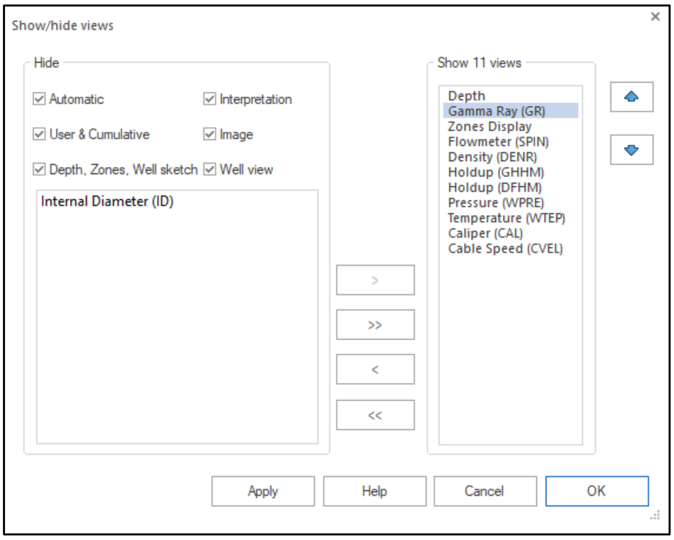
Right click on the GR track, and select 'Show Markers A':



The Marker will only be displayed in the GR track. If you want the markers to be displayed in all the tracks, check the option Show in all views, in the Markers tab of the Well details window.

All the plots can be dragged from the title bar, in order to re-arrange the layout. If a plot is dragged above the plotting area, the plot will be hidden. Another way to manipulate the plots is explained below:

Click on the 'Show', Show, option. The list on the left shows the hidden plots, while the one on the right shows the displayed plots. Click in Internal Diameter (ID), and using < move it to the 'Hide' list. Using the up and down arrows, re-arrange the order of the plots, as shown in the following image:



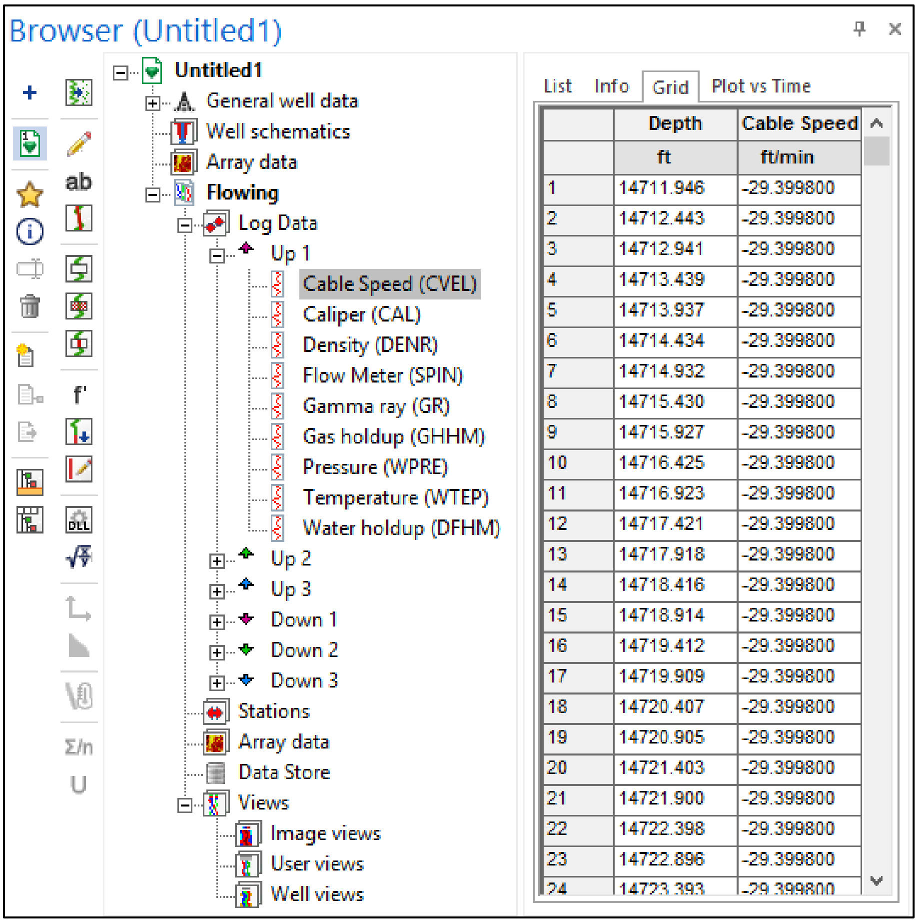
We will save this layout, by taking a snapshot. Click on Snapshots, and then Add. Call this snapshot 'PL Data'.

6. Using the Browser

Click on the ‘Browser’, icon to open the browser dialog.

The window is divided into two areas. On the left is a hierarchical tree view of the data in the file. A node in the tree starting with a ‘+’ can be unfolded with a click on the ‘+’. The window can be attached to any side of the application (always visible or in auto-hide mode), or can be left floating.

A node labelled ‘Log data’ contains all passes. This node in turn unfolds to show the individual pass nodes, which contain the individual channels, or curves. Inside the survey icon also appears a ‘Data Store’ icon. The data store is used to store copies of existing channels for data manipulation, averaging, etc. The right side of the browser contains four tabs: List, Info, Grid, Plot vs Time, used when the selected node in the tree is a channel.



In addition to providing a clear view of the data structure, the browser gives access to most of the Emeraude editing options, also found in the Edit/QAQC tab. Options are selected in the browser toolbar or alternatively with a right mouse button click in the browser window. A quick description of the browser toolbar options can be obtained by moving the mouse successively on top of each button (without clicking).

7. Tool Info

In the 'Tool Information' dialog, characteristics of the tool string used in the survey are entered. A tool is defined for each mnemonic of type pressure, temperature, flowmeter, density, or capacitance. The tool info is a survey property, so if a mnemonic is present in several surveys, it could have different characteristics (blade diameter, capacitance calibration, etc.).

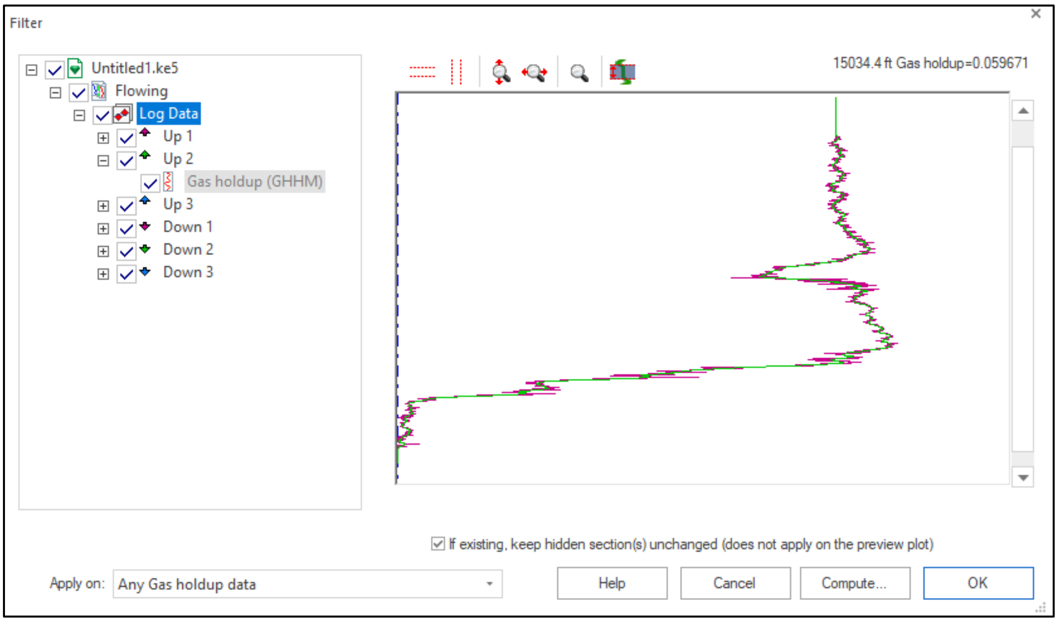
In Tool info, define the density tool DENR as a nuclear tool. Enter the SPIN blade diameter at 3.5''.

8. Edit/QAQC

The Edit/QAQC tab contains a series of Editing options. For the sake of the demonstration, the filter and the derivative option will be used.

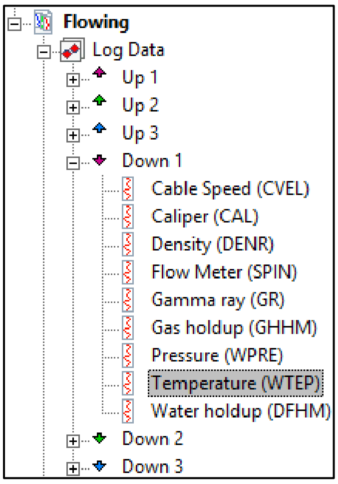
Using the 'Pick nearest', icon, click on the GHHM channel. The exact position of the click is not important. The browser will open, displaying the mnemonic and the pass selected by this option.
Click on Filter.

The Filter dialog displays the selected curve. Clicking on Compute..., the type of filter and the window can be selected. Let's apply an Average filter, with a 4 ft. window. Include all the passes contained in the survey. Exit with OK.



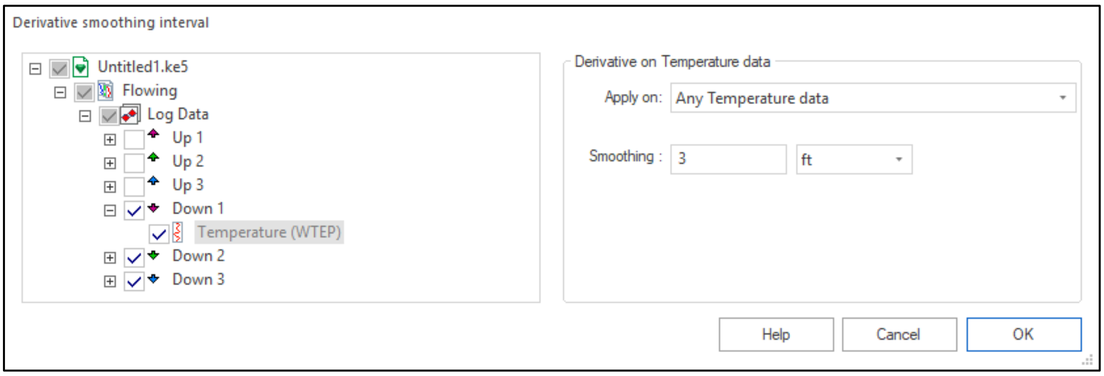
When exiting the Filter dialog with OK, the operation is automatically launched and the corresponding plot is updated. Note that this operation is irreversible.

Now let's create the Temperature derivative, which may help to determine the position of the inflow zones. From the browser, expand node Down 1, and click on the temperature node (WTEP):



Click on f' Derivative in the Edit/QAQC tab. Note also that the editing options are present on the left side of the browser.

Select only the down passes and apply a 3 ft. smoothing:



When clicking on OK, the derivative algorithm is automatically launched, and a new plot is created to display the calculated temperature derivative. The browser now contains a new node dWTEP/dZ, °F/ft (WTEP_D) in the down passes.

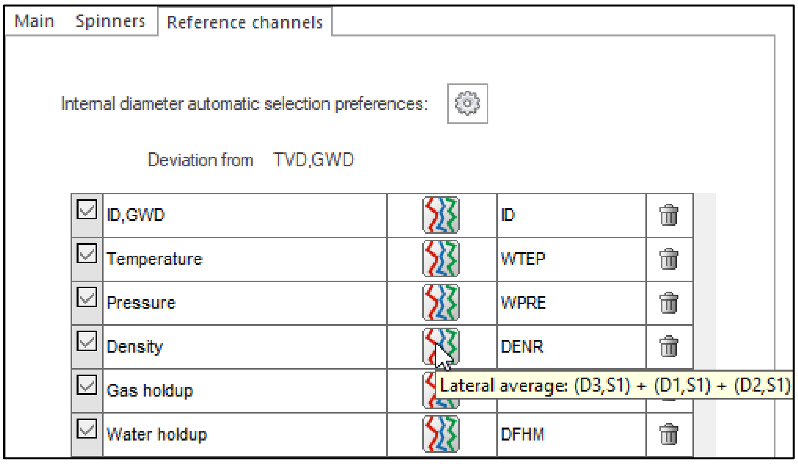
The temperature derivative works as a magnifying glass to amplify the temperature changes, which may be related with fluid inflows.

9. Interpretation

Create a new PL interpretation, by clicking on the 'Info', icon of the Control Panel. Accept the default name. We will start using the Zoned method, with a strict slip model compliance. In the Spinner tab, select the only one available.

Go to Reference Channels tab. Click on the 'Define', icon and select all the down passes for the Temperature, Pressure, Density, Gas Hold-up and Water Hold-up.

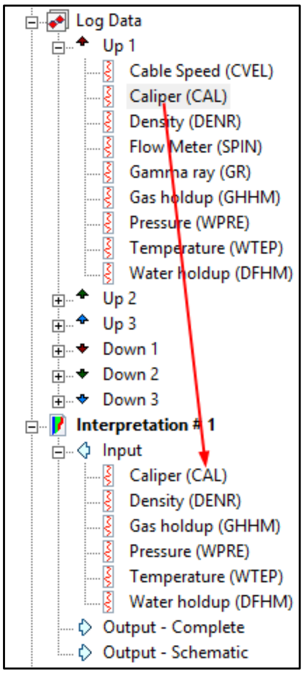
Note that by selecting all the down passes, a lateral average is created. If you hover the mouse over the Define button, you can recall the selection made.



Accepting this window, as the Show reference channel and match views is checked by default, new tracks will be created, showing the Density, water hold-up and gas hold-up 'match' views, and a single track containing all these curves.

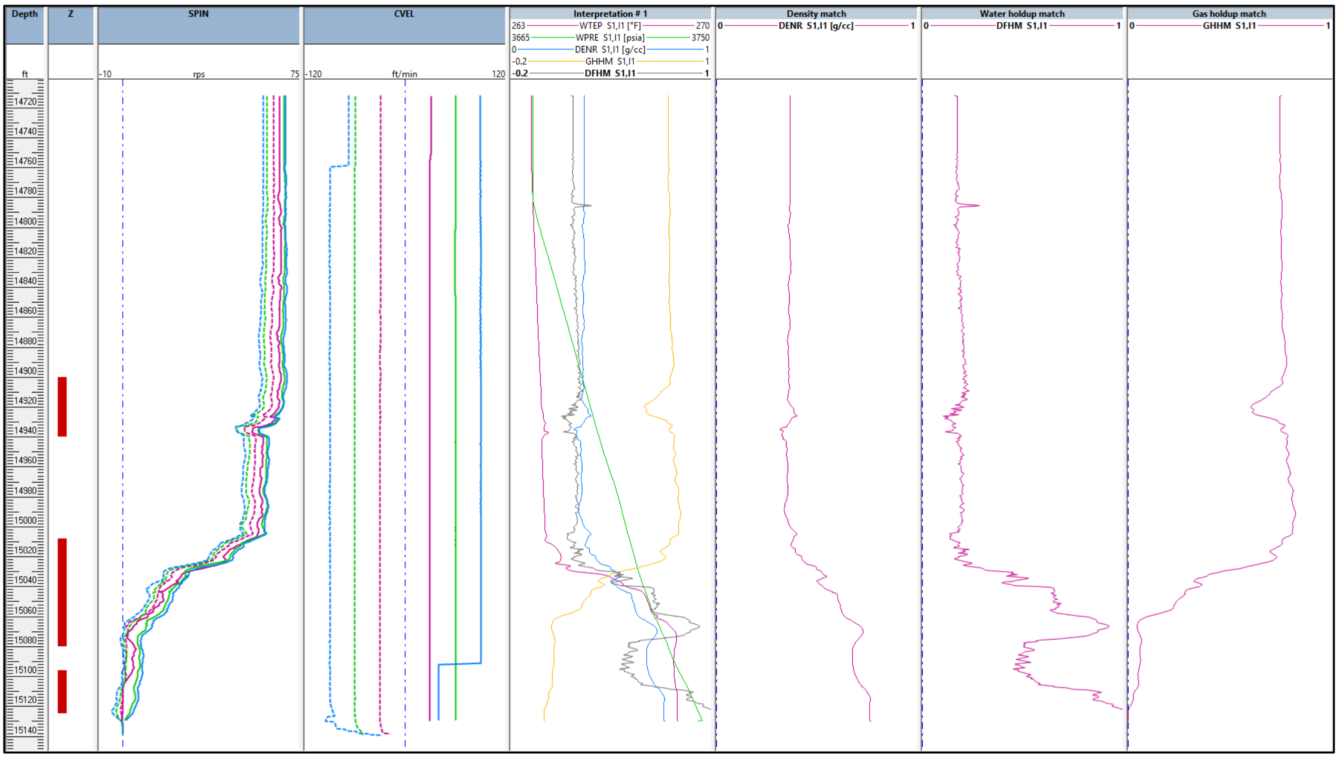
Using the 'Show' button, display the following tracks only: Depth, Zone Display, Flowmeter, Cable Speed, Interpretation #1, Density match, Water Holdup match, Gas holdup match.

In the image above, showing the Input logs, it can be seen that the internal diameter that will be used for the rate calculations comes from the ID loaded before, using the 'Details' icon. The current dataset contains a Caliper, that can be used for the Interpretation. This can be done from the browser, drag and dropping a caliper measurement from a pass to the Input node of the interpretation. We are not going to do this in this example.



We will save this layout, by taking a snapshot. Click on [Snapshots], and then [Add]. Call this snapshot 'Interpretation channels'.

The work space should look like the following picture:



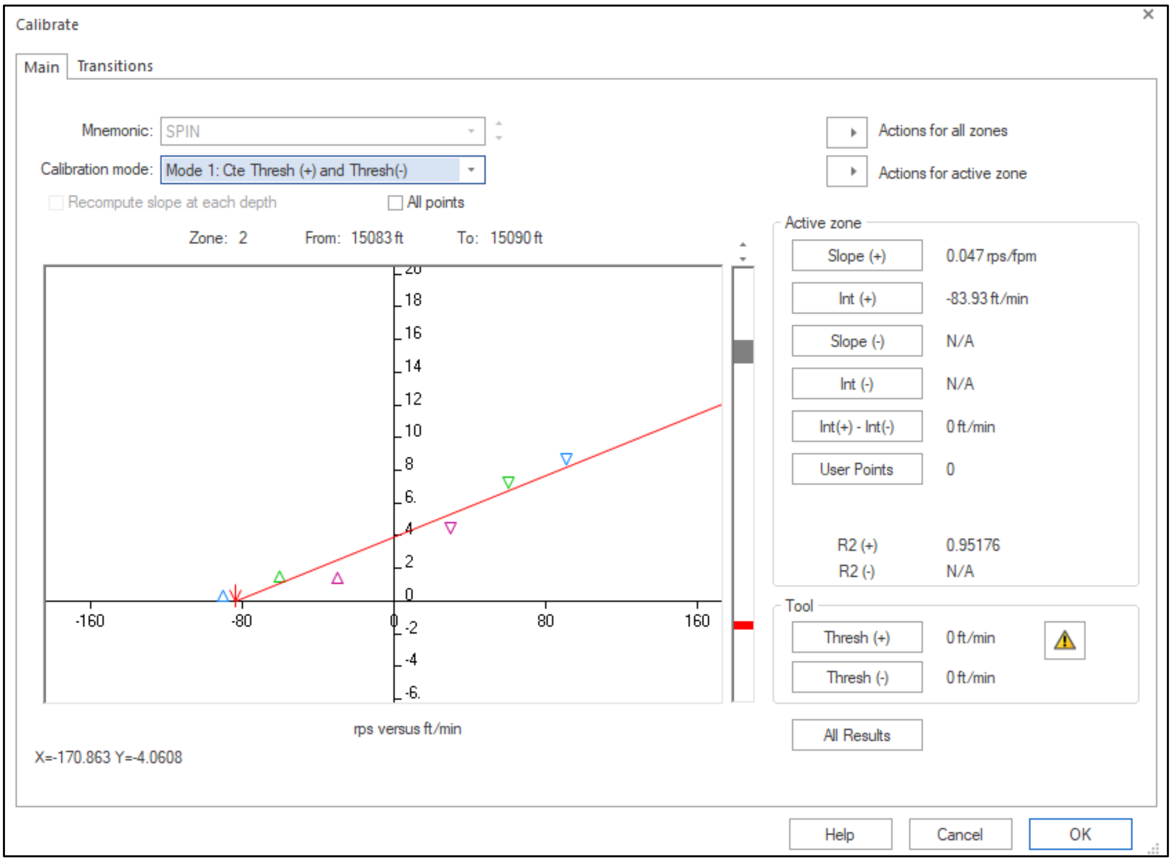
9.1. Spinner Calibration

In order to obtain the apparent fluid velocity from the Spinner rps and Cable Speed data, it is necessary to obtain the slope and the threshold of the spinner in the fluid or fluid mixtures present in the wellbore. This is achieved by plotting this data in an rps vs Cable Speed plot, simply, the Calibration Plot.

Before accessing to the spinner calibration plot, it is necessary to indicate what data will be plotted on it, this is, we need to create the spinner calibration zones. The rps and Cable speed data has to be steady, and also in a constant fluid type.

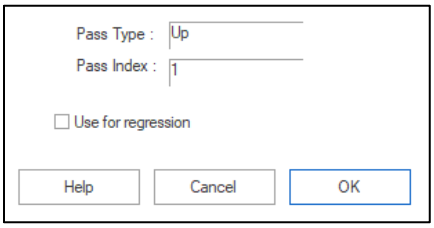
Using the 'Zone grid Editing', icon from the Main tab, insert manually the spinner calibration intervals. Go to the third tab, Calibration, and enter the following: [14845 – 14865 ft.] and [15083 – 15090 ft.].

Click on 'Calibrate'. The Calibration plot shows the two lines corresponding to the different zones. The active one is highlighted in red. To activate a different zone you may click on the vertical schematic, or use the . Right click on the plot window, and a zoom option is available.
Select the bottom calibration zone, and zoom in around the point (right click in the plot to access the zoom options):



The triangles pointing up correspond to up passes, while the ones pointing down are the down passes. The color of the triangles is the same as the colors used in the curves.

Click in the slowest up pass (Up 1 in red) and uncheck the Use for regression box:

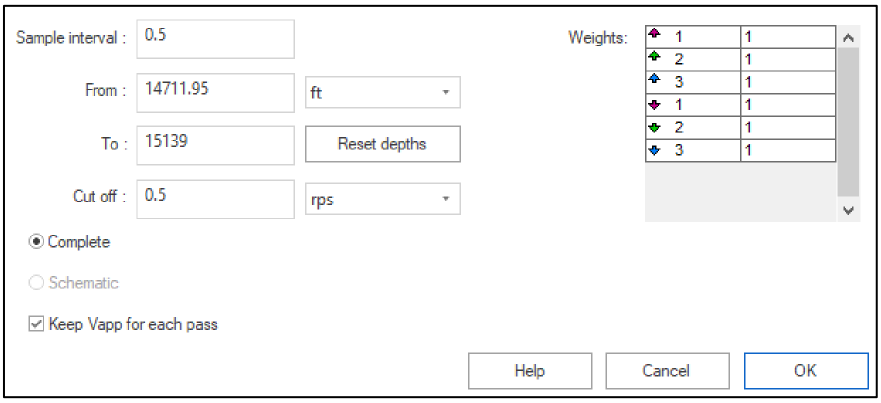


Repeat the same procedure for the Up 3 pass. The line is automatically recomputed when a point is added or removed.

In this dataset it's not possible to find the spinner threshold, as there is no clear no flow zone. A value of +/- 3 ft./min will be used for both calibration zones. Therefore, we can use Calibration Mode 1: Mode 1: Cte Thresh (+) and Thresh(-)

Click on Thresh (+) and enter a value of 3 ft./min. Repeat the same with Thresh (-), using a value of -3 ft./min.

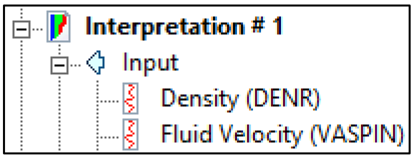
We will skip the Transitions tab now, and will proceed with the generation of the apparent velocity. Clicking on OK, this process is automatically launched. We will use all the passes, so the weight remains 1. Set a cut off of 0.5 rps, to avoid the non-linear region of calibration. Also, make sure that the 'Keep Vapp for each pass' is checked.



Click on OK.

Two new tracks are created, one showing the Vapp for each of the selected curves (VASPIN), and the Velocity match track, showing the average of all the VASPIN curves. The apparent velocity, and therefore these tracks will be automatically updated every time the Vapp generation is launched. We will save this current Vapp to compare it with a new one, adding Transitions between the calibration zones.

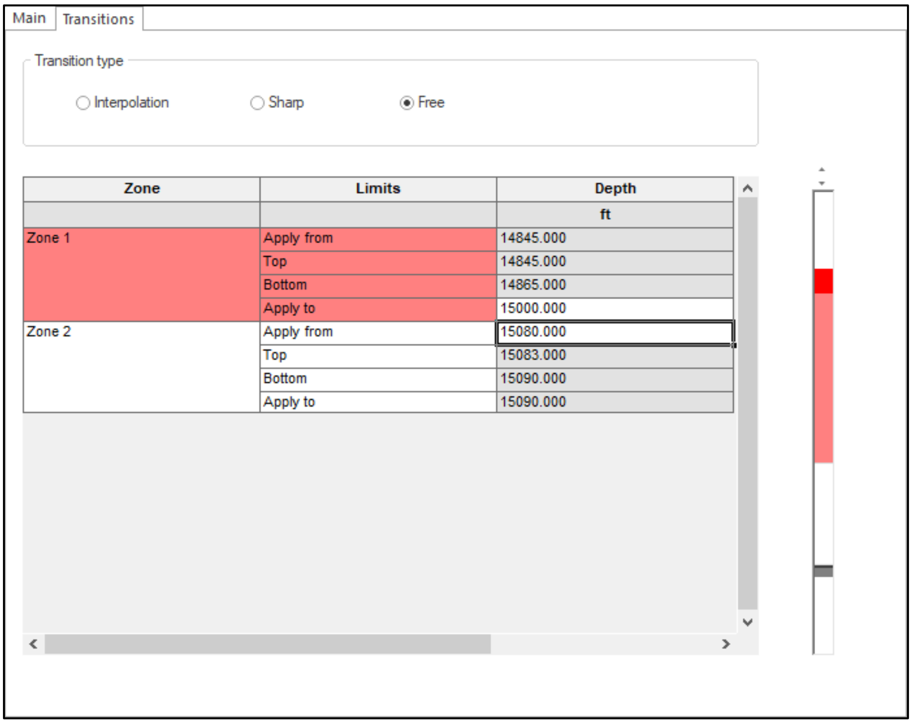
Open the 'Browser'. Under Interpretation input, the Fluid Velocity (VASPIN) corresponds to the current Apparent Velocity.



Drag and Drop the Fluid Velocity under Data Store

We will define the spinner calibration application zones. The slope and thresholds found in the calibration plot corresponds only to the fluid where the spinner was immersed, in that specific calibration zone. In this survey, the fluid changes between the upper and the lower calibration zones, however, no stable calibration intervals were found.

Go back to Calibrate, and second tab 'Transitions'. Select the 'Free' transition type, and set the Apply to and Apply from, as shown in the picture below:

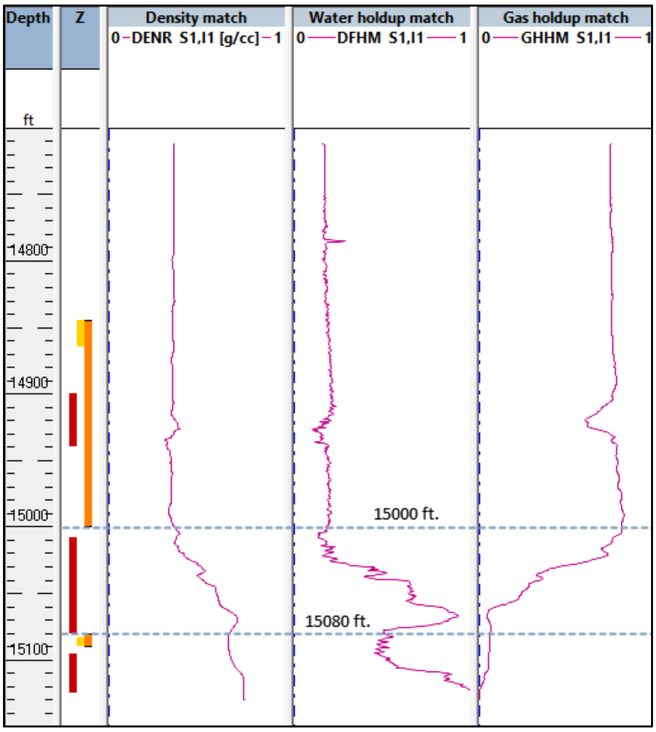


Zone	Limits	Depth
		ft
Zone 1	Apply from	14845.000
	Top	14845.000
	Bottom	14865.000
	Apply to	15000.000
Zone 2	Apply from	15080.000
	Top	15083.000
	Bottom	15090.000
	Apply to	15090.000

Exit with OK, apply to all and generate the apparent velocity with all the defaults.

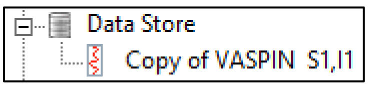
In the Zone display track (Z), the application zones are shown in orange. The calibration parameters found from the upper calibration zone will be used from the top of the logged interval, to the bottom of the orange zone. This depth was selected because the density, water hold-up and gas hold-up remain fairly constant.

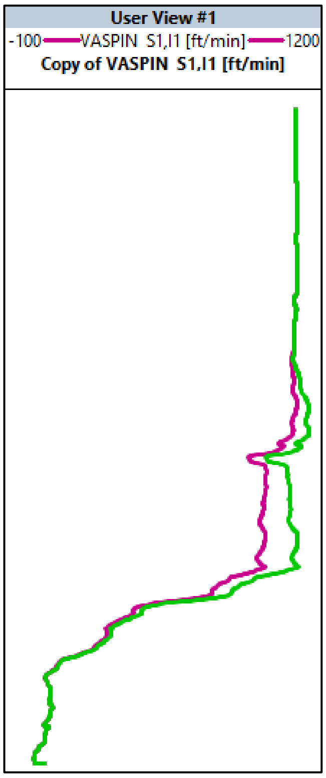
The bottom calibration parameters will apply from the bottom of the logged interval to the top of the lower orange zone. In between the two application zones, an interpolation of the calibration parameters will be carried out.



A new fluid velocity is generated and the Velocity match track is updated. We will compare it to the previous one.

From the drop down list of the 'New', icon, select User view. A blank track is added on the right of the workspace. Open the browser, and drag and drop the Fluid velocity curve under Input, and the one saved in the data Store.

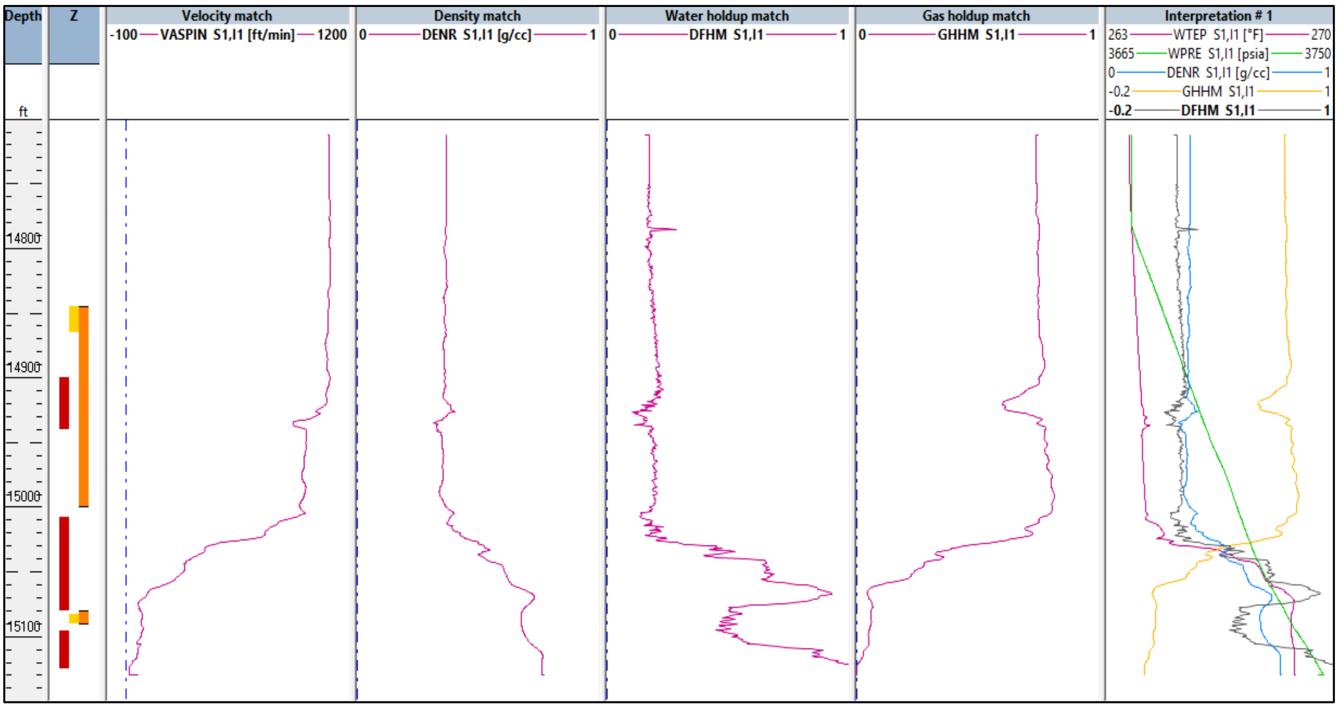




Comparing the 2 fluid velocity curves, it's clear that the calibration application zone definition will greatly affect the fluid velocity shape, and therefore the calculation of the rates and contributions.

We will use the last fluid velocity generated.

At this stage, we don't use the rps and cable speed data any more. Clean the work space so it looks like the image below, and update the 'Interpretation channels' snapshot.



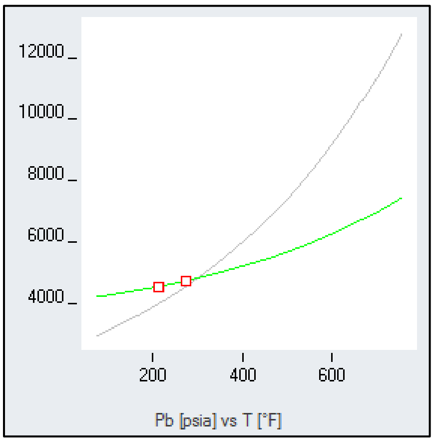
9.2. PVT

Before starting with the Rate calculation process, Emeraude needs a PVT model.

When the PVT model is created or changed, the definition of each phase's properties has to be performed sequentially from left to right using the corresponding toolbar. For each phase dialog, input data are entered and correlations are selected in the first page; each additional page is assigned a property.

In the PVT window define the fluid type as saturated oil with water. Enter 1100 cf/bbl for the solution GOR, Water salinity = 2000 ppm, gas SG = 0.786, oil gravity = 35.5° API.

The fluid modeling consists on a black oil PVT, which is initialized by the inputs mentioned above and correlations. These correlations can be constrained with Lab PVT data.
In the Oil window , click on the Pb tab. Then click on Constraints , and add the following Temperature vs Bubble point values: [205 F, 4650.4 psia], [265 F – 4832.6 psia]. Accept with OK, and Match constraints .
The plot shows how the actual Lab data values are used to constraint the PVT correlations:



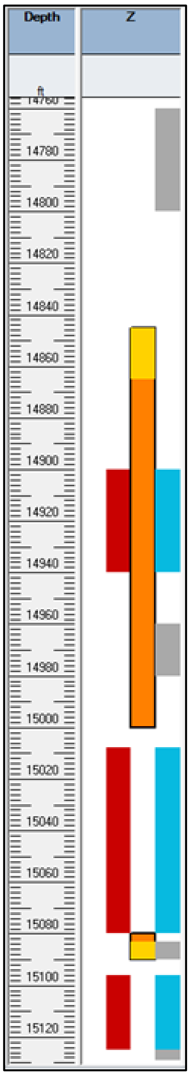
9.3. Rate Calculation

Having defined all the necessary input logs, and with the PVT model created, we are ready to run the rate calculations. We will explore the different methods that Emeraude offers: Zoned (with Strict and Loose slip compliance) and Continuous. Will start with Zoned – Strict slip compliance.

9.3.1. Zoned method – Strict slip compliance

In the zoned method the rates are calculated only at the Rate calculation zones. As a Strict slip compliance is used, the holdups are dictated by the slippage correlations. First, we need to define the rate calculation zones, to be selected in regions of stable fluid velocity and holdups, and also above and below any perforation.

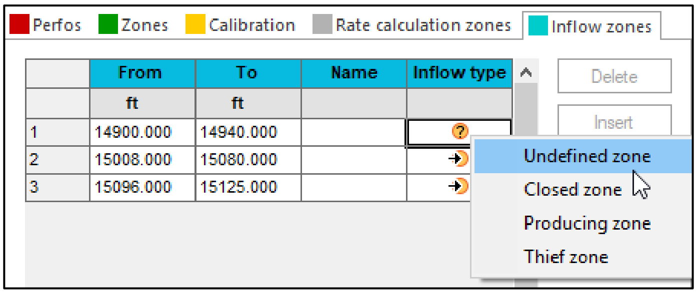
Click on the 'Edit', , icon and go to the fourth tab, 'Rate calculation zones'.
Enter the following: [14760 – 14800 ft.], [14960 – 14980 ft.], [15083 – 15090 ft.] and [15125 – 15129 ft.].
Click on Apply to validate.



By default, between 2 Rate calculation zones, one inflow zone is created (in blue). As the perforations depth was informed, the Inflow zones are of the same size than the perforations. However, if no perforations were defined, the inflow zones will occupy the full gap between the rate calculation zones.

Note that the perforation may not be producing homogenously from the full interval. Therefore, the size of the inflow zone may be modified to account for the actual contributing length. Later we will analyze whether this is necessary or not.

In the fifth tab of the Edit button, the Inflow zones are defined. Set (right click) the inflow type of the three zones to Producing:



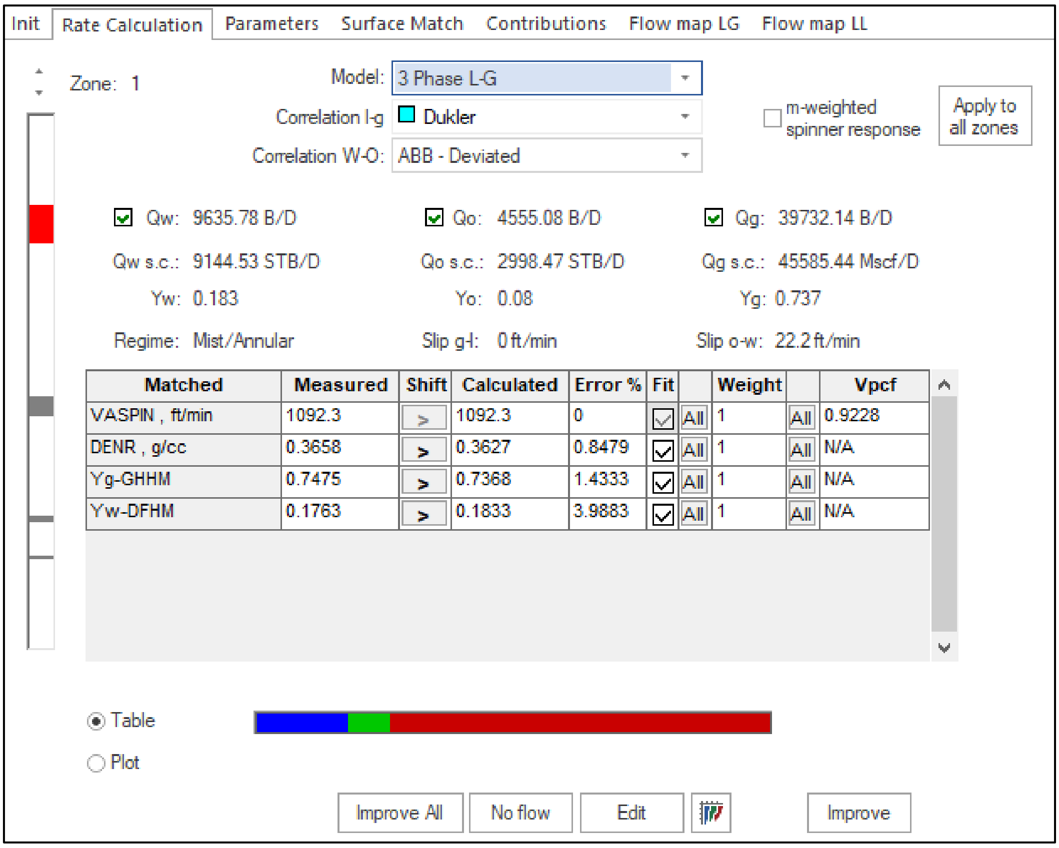
Now that the rate calculation zones are defined, we can proceed with the actual rate calculation.

Clicking on the 'Rates' , icon the 'initialize rate calculation zones' dialog pops-up.
In the first tab, Init, the 3 Phase L-G model is proposed, based on the PVT and reference channels. We will use this model, with the suggested correlations.

In the Rate Calculation tab, the values displayed in the dialog correspond to a given calculation zone. Similar to the spinner calibration dialog, a small schematic shows the active zone and can be used to change the zone under investigation.

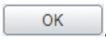
To move to another zone, click directly on the zone in gray, or use .

The measured and calculated values (shown in the 'match' views) are also presented, with the percentage of error between them. If certain measurement is not to be used, for example due to bad data quality, remove the check from . The weight of the measurement on the non-linear regression can also be controlled from here.

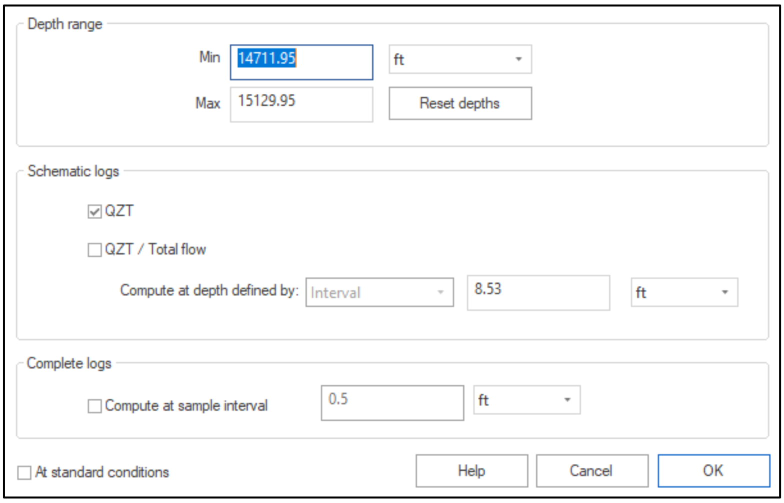


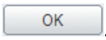
To summarize, for any assumption of the rates at a given depth, and any flow correlation, we can calculate the simulated apparent velocity, and the simulated density and holdups. Emeraude can then run a non-linear regression in order to match measured and simulated values. The displayed values of rates result from this calculation.

Move to the bottom zone (Zone 4) with . This zone is below the bottom most perforation zone, so it should show zero rate. Due to a non-perfect spinner calibration, this is not the case, and some rate is predicted from here. Click on No flow to force this zone to zero rate (make sure that you are in zone 4).

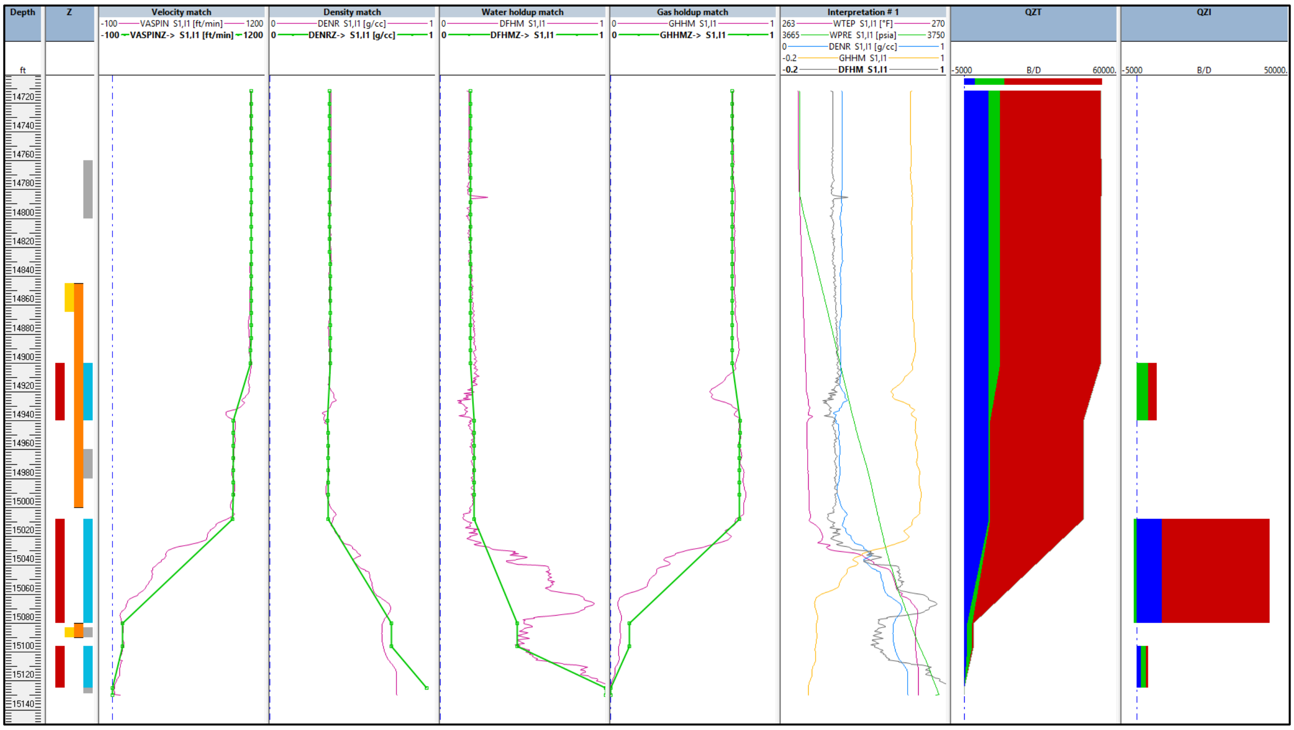
Click on 

. The Rate Log setting window is automatically launched:



Accept with 

.



Two new plots were created: QZT showing the cumulative Production, and QZI showing the contributions from the different inflow zones. The 'match' views now show the simulated values, as well as the measured values.

On the top of QZT plot, a bar shows the surface reported contributions, taken to downhole rates using the volume factors. In this case, there is a good agreement between the PLT calculated and the surface reported rates.

Couple of things can be seen from this view:

- The simulated velocity doesn't match the calculated one at the inflow zone level. The simulated velocity varies linearly between the top and the bottom of the inflow zone, which in this case are the same size of the perforation zones. As the contributions are not homogeneous along the perforation, this can be corrected refining the inflow zones.
- The holdups and density don't match perfectly. To solve an N phase case, we need N-1 hold-up/density tools. In this case, there are more than 2 measurements for the 3 phases, so the problem is overspecified. Emeraude can handle this situation, but as the physics of the measurements is different, the error induced will be higher. The relative weights assigned in the regression on the measurements will of course drive the solution towards one side or the other.
- Maximizing the QZI plot, it can be seen that the middle zone is producing water and gas, while injecting oil. This is not physically possible, and it's called a 'downhole separator'. This is a consequence of calculating the rates independently in each rate calculation zone. The Global Regression can be used to resolve inconsistencies of this type when they arise, by imposing additional constraints to the solution.

To help solving the first situation we will reduce the inflows length of the top and bottom perf, and subdivide the middle perf in small inflow zones. Go to the Edit button, fourth tab 'Rate calculation zones', and copy the values below:

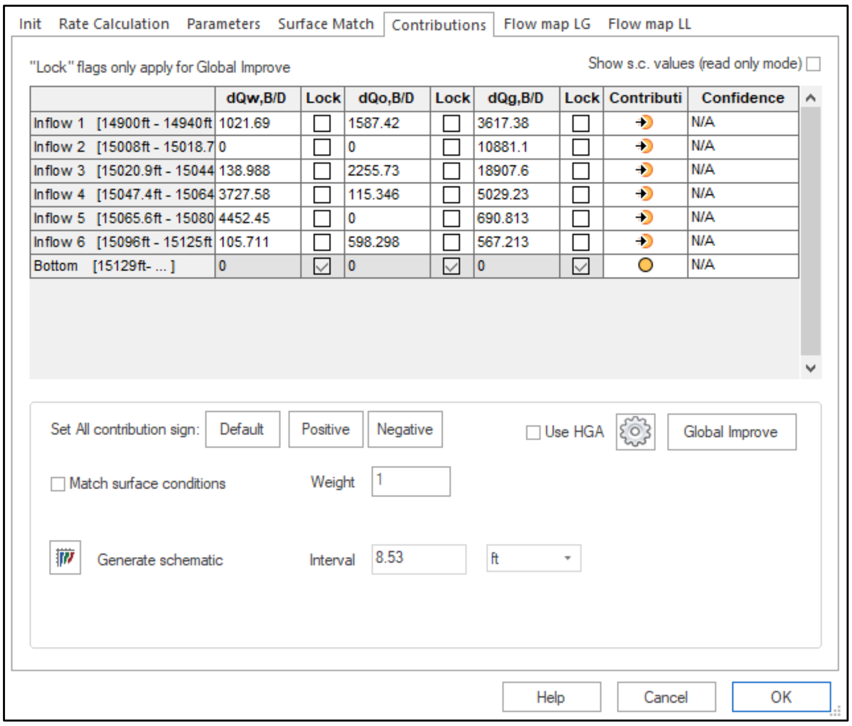
Perfos | Zones | Calibration | Rate calculation zones | Inflow zones

	From	To
	ft	ft
1	14790.700	14809.100
2	14952.900	14965.400
3	15018.700	15020.900
4	15044.800	15047.400
5	15064.300	15065.600
6	15083.500	15090.800
7	15125.300	15129.000

Delete | Insert | Add

New inflow zones are created in the middle perforation. Go to the Inflow zones tab, and modify these inflow zones.

Now, let’s go back to Rates, ‘Contributions’ tab:

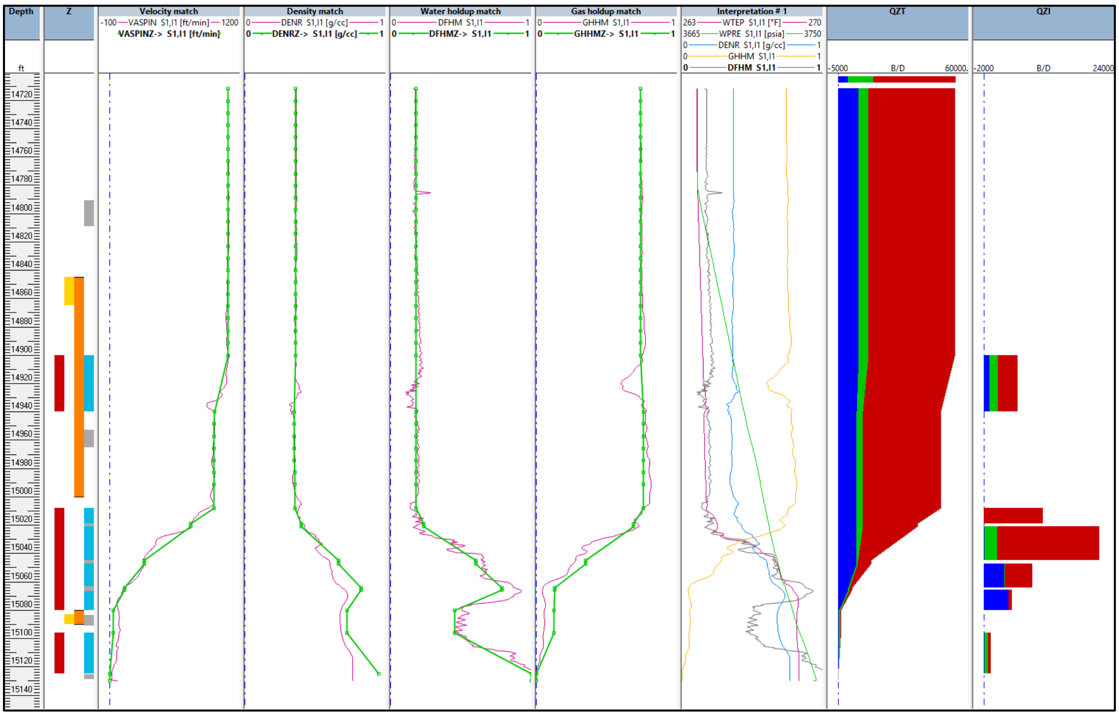


The dialog displays the list of the inflow zones, and the contribution for each phase. All values in the dialog are editable; if a value is changed the cumulative rates are recalculated. If you change a value before running ‘Global Improve’, the value can be fixed by checking the lock checkbox. Contributions from the well bottom are zero and they do not need to be taken as variables. They are already locked.

The contributions of the different inflow zones are displayed. Note the positive and negative signs, indicating downhole separators. Set All contribution sign to Positive, and then Global Improve.

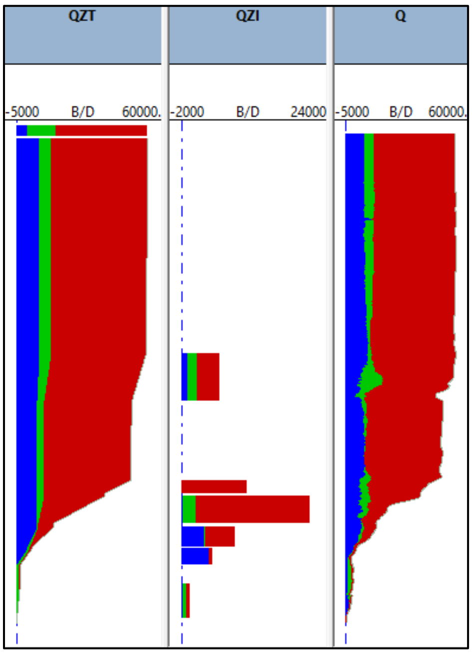
The Global Regression starts. Values inside the grid are updated after each iteration. This time, all zones are treated simultaneously and the unknowns are the contributions. The goal is not only to match the input measurements, but also to satisfy the defined constraints. When the regression is over, a text at the bottom of the dialog gives the value of the objective function, an indication of the match quality (the lower, the better). It can be seen that the sign constraint has been honored and the Global regression ends up with the best possible match, given those constraints and the input measurements. As in the first step of the diagnosis, it is important to realize that the Global regression only worries about the error on the calculation zones. The match outside of those zones is simply not considered. Therefore, everything relies on the position of the calculation zones, and the stability of the data in those zones.

The generated log should look as the image below, with all the contributions positive.



Click on the 'Logs', , icon. The Rate logs settings are controlled from this dialog. At the bottom, the ☐ At standard conditions box offers the possibility of plotting the QZT and QZI at standard conditions. In this case, this won't be very convenient as the gas will expand and the oil and water won't be visible. Leave it unchecked. Check the ☑ Compute at sample interval box, to compute the Complete logs.

A new track, 'Q', is generated. In the Complete log calculation, the same non-linear regression scheme as used in Zone Rates is applied at every depth specified by the user.

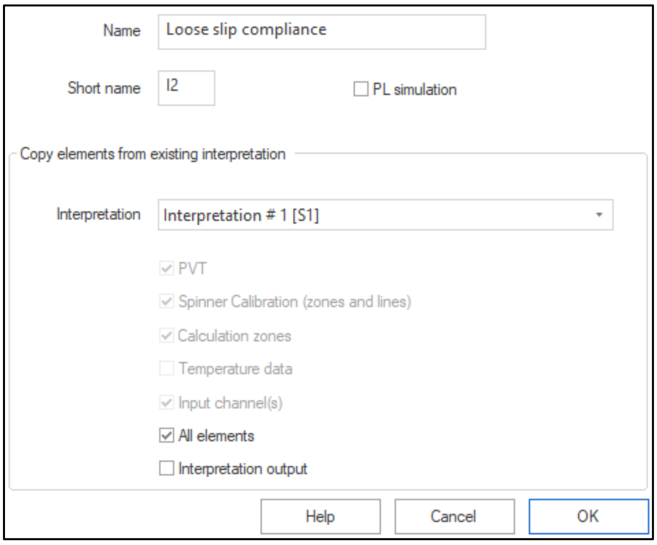


In addition to the rate channels, when selecting the Complete log calculation, Emeraude also outputs a mixture velocity, a corrected density channel (if a gradio is used) and a hold-up channel for each phase. This can be found under Output - Complete, in the browser.

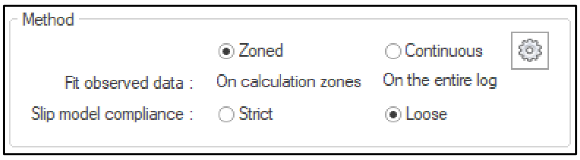
This ends the interpretation using the Zoned method, and strict slip compliance. Let's apply a different calculation method.

9.3.2. Zone method – Loose slip compliance

From the interpretation tool strip, click on New interpretation. Call this interpretation 'Loose slip compliance' and copy all the elements:



The Interpretation settings window pops up, with a copy of the previous interpretation settings and reference channels. Modify the slip model compliance to Loose:



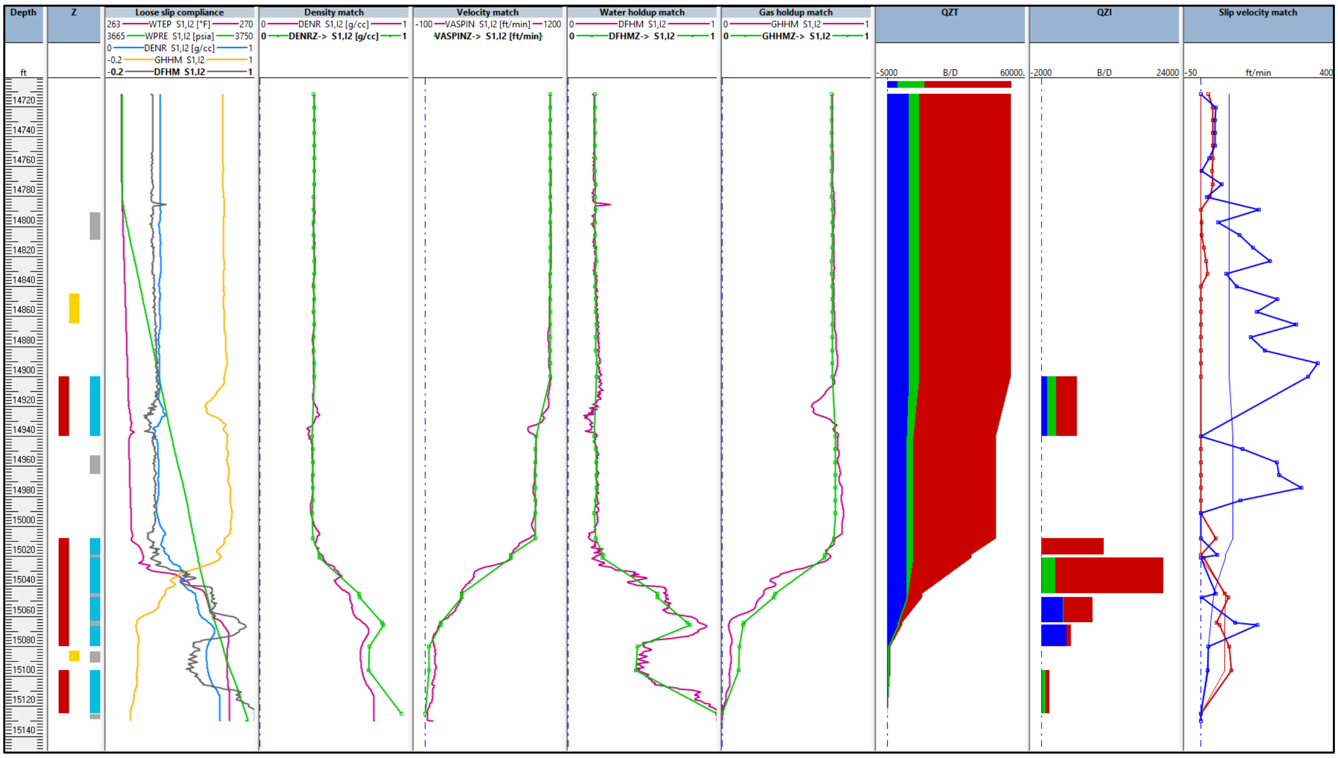
Accept with OK.

Go back to Rates. The table now shows that the hold-ups are variables of the regression, and not dictated by the slippage correlations as before:

Matched	Measured	Shift	Calculated	Error %	Fit		Weight		Vpcf
VASPIN , ft/min	1091.22	>	1091.23	0	☑	All	1	All	0.9228
DENR , g/cc	0.3623	>	0.3599	0.6526	☑	All	1	All	N/A
Yg-GHHM	0.7489	>	0.7405	1.1231	☑	All	1	All	N/A
Yw-DFHM	0.1742	>	0.1797	3.1824	☑	All	1	All	N/A
Yw	0.1797	>	0.1797	0	☑	All	1	All	N/A
Yg	0.7405	>	0.7405	0	☑	All	1	All	N/A

Go to the Contributions tab, and set All contribution sign to Positive, and then Global Improve.

Exiting with OK the plots are updated, and a new track is generated:



As the Loose slip compliance method allows the holdups to differ from the slip model holdup prediction, the slip velocity match track shows the difference between the slip model predictions (solid lines) and the slip velocities used by Emeraude (lines with markers) for liquid-gas (in red) and for oil-water (in blue). Only the deviation from the slip models allows matching closely the holdups and the density.

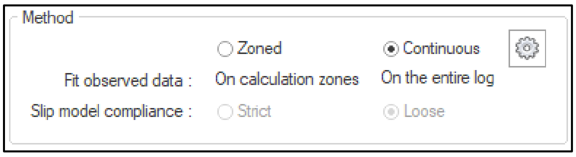
Note that in this case, the calculations are still at the rate calculation depth. To consider the full data, let’s investigate the last type of interpretation method.

9.3.3. Continuous method

The 'Continuous' method differs from the 'Zoned' method in seeking in the Global Regression an agreement between the schematic logs and the data everywhere, and not just on the calculation zones. If we look at the previous plots obtained, we see that schematics are … schematic, i.e. they do not show much variation outside inflow zones. This is linked to the fact that holdups are controlled by the slip model predictions. In order to give the schematics the ability to reproduce the local variations seen on the data, the 'Continuous' method treats the holdups as variables (in addition to the rates) but constrains them by the slip model holdup predictions. In other words, the holdups are freed but the process tries 'not to go too far' from the slip model prediction.

From the interpretation tool strip, click on New interpretation. Call this interpretation 'Continuous' and copy all the elements.

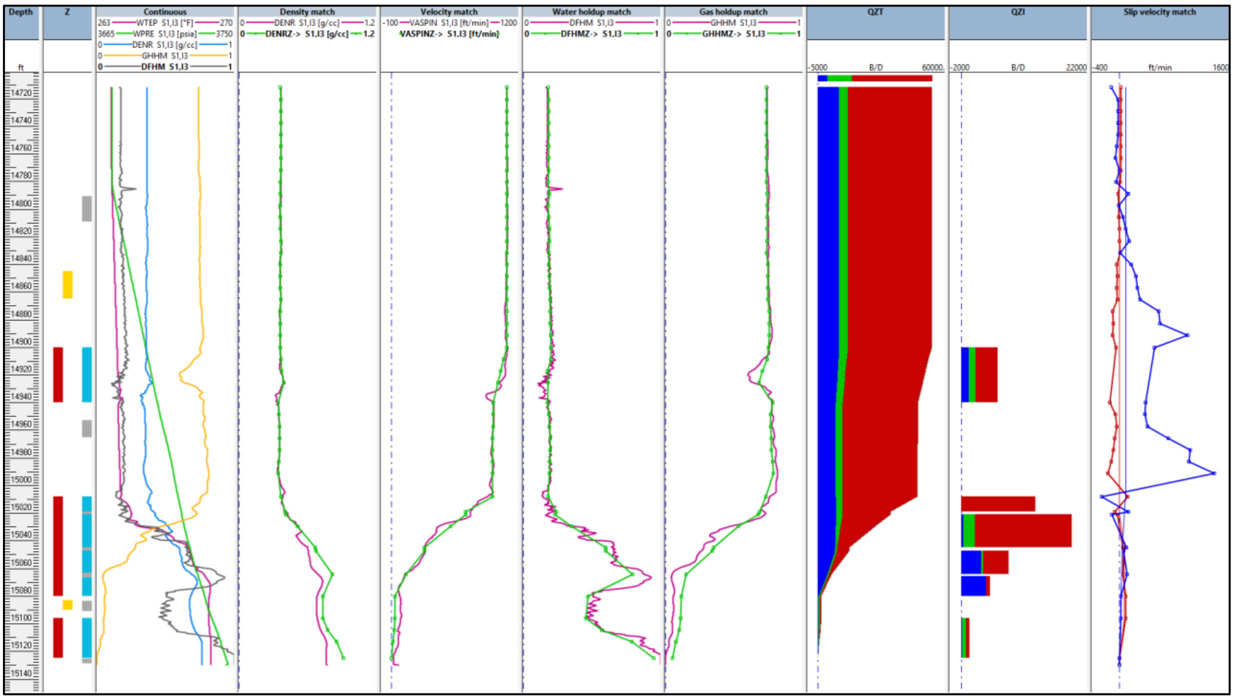
In the interpretation settings dialog, now change the interpretation method to Continuous:



'Rates' is renamed 'Inflows' to emphasize that the 'Continuous' method is really about solving at once for the inflows, rather than solving the zones individually. The dialog takes you directly to the 'Contributions' tab.

Set All contribution sign to Positive, Uncheck the 'Constrain slippage sign' box, and then Global Improve.

The schematics are updated after each iteration and the successive rates are seen on the QZT and QZI tracks. Finally, validate with OK.



We have applied three different interpretation methods to this dataset.

The loose slip compliance in zone method should bring no real benefit here; it would only if we had some phase velocities or rates, something to match against that would implicitly determine the slips. In fact, we see that there are only very small differences between the two-zoned interpretations.

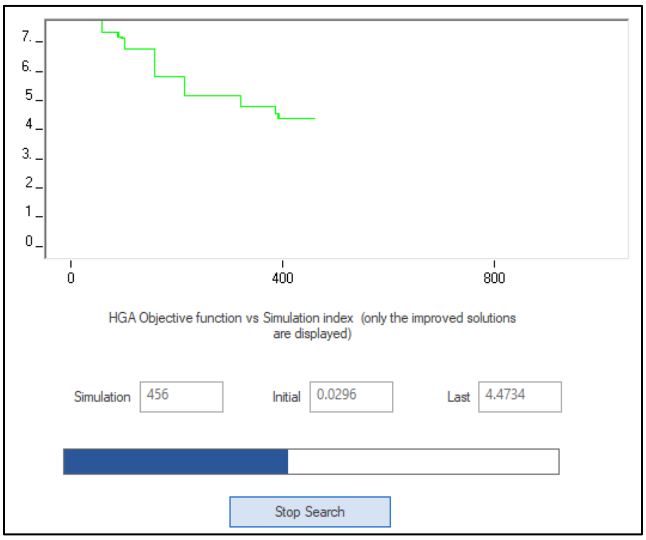
The continuous method can bring more confidence by matching the global picture better. Again, in the absence of additional constraints or measurements that would 'auto-determine' slips, the picture will not be affected much, as we see. Of course with other measurements, or incorporating Temperature, things would be different.

9.3.4. Hybrid Genetic Algorithm (HGA)

It is possible to couple the Global Regression with a pre-processing called HGA, for 'Hybrid Genetic Algorithm'. This pre-processing is particularly suited at avoiding local traps or finding a better starting point for the regression. In the contributions tab, check the Use HGA box.

The button to view/edit the HGA parameters.

When running a Global Regression with the HGA option checked, a dialog appears to show the HGA progress:



HGA is not 'biased' by the starting point and it will usually start with solutions having a much higher (worse) objective functions than the base case. Also, a Genetic Algorithm improves the solution iteratively, but unlike a classical optimization, it does not provide a systematic improvement at each iteration. This will be translated in the plot where new points are drawn only when there is an improvement. So the plot may seem to be halted for some time and then start again. An automatic scaling is executed every 5 new points.

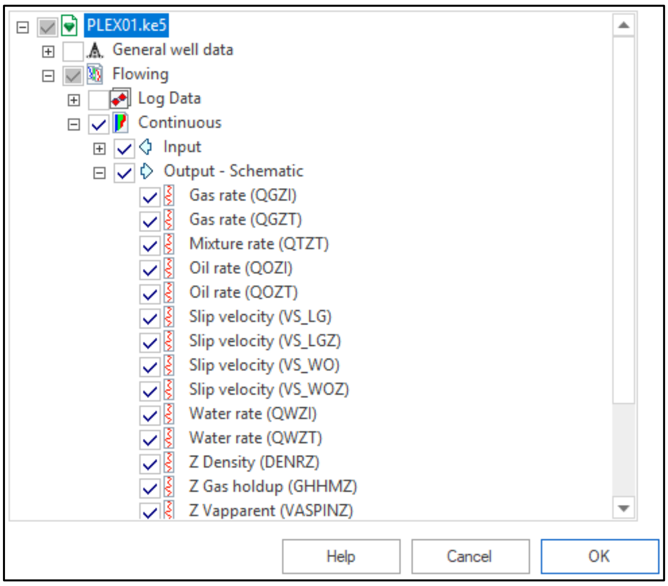
If HGA finds a solution with an objective function smaller than the base case, then after completion or termination, this solution will be used as the new starting point of the Global Regression. If HGA stops without such a better solution, the Global Regression starts from the base case. *Note that you can stop HGA at any point, in particular if you see that it successfully reduced the objective function.*

10. Output

Moving to the Output tab, a number of ways to display the results are presented.

Click on the ‘Table’, , icon. The results of the active interpretation of the current survey are reported at the inflow zone level.

Click on the ‘Export File to folder’, , icon. Any pass or interpretation curve can be exported in LAS, LIS or ASCII format. The image below shows how to export the Inputs and outputs of the Continuous interpretation:



This tab includes features to create quick reports.

This concludes Emeraude (CHL) Tutorial #1.