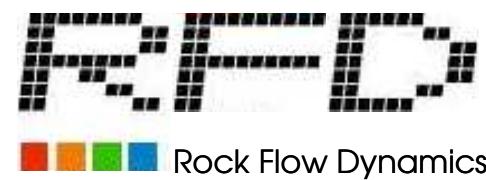


tNavigator® 17.3

Rock Flow Dynamics



Simulation User Guide



September 2017

Copyright Notice

Rock Flow Dynamics® (RFD), 2004-2017. All rights reserved. This document is the intellectual property of RFD. It is not allowed to copy this document, to store it in an information retrieval system, distribute, translate and retransmit in any form or by any means, electronic or mechanical, in whole or in part, without the prior written consent of RFD.

Trade Mark

RFD, the RFD logotype and tNavigator® product, and other words or symbols used to identify the products and services described herein are trademarks, trade names or service marks of RFD. It is not allowed to imitate, use, copy trademarks, in whole or in part, without the prior written consent of the RFD. A graphical design, icons and other elements of design may be trademarks and/or trade dress of RFD and are not allowed to use, copy or imitate, in whole or in part, without the prior written consent of the RFD. Other company, product, and service names are the properties of their respective owners.

Security Notice

The software's specifications suggested by RFD are recommendations and do not limit the configurations that may be used to operate the software. It is recommended to operate the software in a secure environment whether such software is operated on a single system or across a network. A software's user is responsible for configuring and maintaining networks and/or system(s) in a secure manner. If you have any questions about security requirements for the software, please contact your local RFD representative.

Disclaimer

The information contained in this document is subject to change without notice and should not be construed as a commitment by RFD. RFD assumes no responsibility for any error that may appear in this manual. Some states or jurisdictions do not allow disclaimer of expressed or implied warranties in certain transactions; therefore, this statement may not apply to you. Since the software, which is described in the present document is constantly improved, you may find descriptions based on previous versions of the software.

Press to open tNavigator Library

Press to open User Manual

Contents

Contents	3
1. Introduction	10
2. tNavigator documentation	11
3. Open model in tNavigator	13
3.1. tNavigator main window	13
3.2. Menu bar	13
3.3. Main toolbar	16
3.4. Module's Tabs	16
3.5. tNavigator window with opened model	20
3.6. Document. View. Files. Reports	22
3.7. (Status) Report Panel – log	27
3.8. Top Panel Buttons	29
3.8.1. Create New View	29
3.8.2. Show All Views	30
3.8.3. Hide All Views	30
3.8.4. Save Model	30
3.8.5. Split model	32
3.8.6. History Matching Variables Manager	32
3.8.7. Reload Model	33
3.8.8. Run calculation	33
3.8.9. Playback Results	34
3.8.10. Stop	34
3.9. Left panel buttons	35
3.10. Run model	36
3.10.1. Compute to a fixed step	36
3.10.2. Re-Run Calculations	36
3.10.3. Start Calculations from Any Previously Calculated Step	36
3.11. tNavigator hotkeys	37
3.12. Definitions	38
3.13. Schedule	39

4. Grid Properties	41
4.1. Initial	43
4.2. Calculated	48
4.2.1. 2D maps for Saturation Ternary Diagram	55
4.2.2. Request for the distribution of blocks with convergence problems	57
4.2.3. Request for distributions of total flows of water, oil and gas	58
4.3. Fluid-in-place	59
4.4. Regions	60
4.5. User Cuts, User Maps	62
4.6. Vector Fields	63
4.7. Interblock Flows	64
5. Grid Properties. Right panel buttons	65
5.1. Views and Presentation Parameters	65
5.2. 3D Slicing	66
5.3. Create a Profile	69
5.4. Distance between two specified blocks	71
5.5. Create a Slice Filter	72
5.6. Create a Cross-Section	74
5.7. Export	80
5.8. Well, Groups and Network Filter. Stream Line Filter	81
5.8.1. Well filter	81
5.8.2. Streamline Filter	83
5.8.3. Group filter	84
5.8.4. Network filter	84
5.9. Create Screenshot	86
5.10. Well Actions	88
5.11. Find a Well or Connection	89
5.12. Statistics	90
5.13. Well Selection	92
6. Grid Properties. General principles	94
6.1. Palette	97
6.2. Local Grid Refinements (LGR)	102
6.3. Properties for dual porosity model	103
6.4. Properties in 3D	104
6.5. Properties in 2D	108
6.6. Working with user polygons (contours)	113
6.6.1. Import user polygons	113
6.6.2. How to work with region created by polygon	115
6.7. Bubble maps	116
6.7.1. Visualization settings	118
6.7.2. Bubble Map State	119
6.7.3. Accumulated Bubble Map	120

6.7.4. Correlation Coefficients	121
6.7.5. Mismatch Map	122
6.7.6. Custom	123
6.8. Network visualization	126
6.9. Histogram	128
6.10. Export of grid properties	130
7. Graphs. General principles	138
7.1. Graphs Right Panel Buttons	140
7.2. Select Object and Parameter	145
7.2.1. Search well in a list	147
7.2.2. Sort wells in a list	148
7.3. Well Status	149
7.4. Graph View	151
7.4.1. Scaling along X and Y Axes	151
7.4.2. Expanding Graph Regions	151
7.4.3. Dragging	151
7.4.4. Time Display in the Graph and in the Table	151
7.4.5. Graph View. Change Graph Color and Line	151
7.5. Trend line	153
7.6. Auto sync	155
7.7. Coordinate Axes. Multiple Coordinate Systems in the Same Window	156
7.8. Setting Minimum and Maximum Values for Graph Axes	157
7.9. Graph tables	159
7.10. Multiple Models' Results Graphs in the Same Window	160
7.11. Graph Types (Object Item, Parameter, Step)	162
7.12. Graph templates	166
8. Graphs. Graphs list	168
8.1. Rates	168
8.2. Totals	178
8.3. Fluid-in-place	184
8.4. Analytics	187
8.5. Pressure	197
8.6. Flow Between FIPs	203
8.7. Run Statistics	204
8.8. Crossplots	207
8.9. Well profile	210
8.10. Well section	214
8.10.1. Visualization of RFT (MDT)	215
8.10.2. Visualization of PLT	215
8.11. User Arithmetics	217
8.12. Block Info	220
8.12.1. Request of distributions of relative permeabilities and capillary pressures	221

8.13. Profile info	223
8.14. Pressure/Temperature Slices	225
8.15. Historical vs. Calculated (Hist vs Calc)	227
8.16. Unified History Matching Results	230
8.17. Comparison of Results	234
8.18. Well RFT Mismatch Table	236
8.19. Well PLT Oil Mismatch Table	237
8.20. Tracers	238
8.21. User selection	239
8.22. Aquifers	240
9. Load Well Data	243
9.1. Layers	243
9.2. Trajectories	244
9.2.1. GWTD	244
9.2.2. Trajectory	244
9.2.3. LAS	245
9.2.4. Generalized	246
9.2.5. Dip-circle	247
9.2.6. WellHead	247
9.3. Groups	249
9.3.1. Well – Group	249
9.3.2. Group – Wells	249
9.3.3. Group – Parent Group	249
9.4. Events	250
9.5. Well History	252
9.5.1. History table	252
9.5.2. History – FHF Format	255
9.6. Well Logs	256
9.6.1. Well Logs (LAS)	256
9.6.2. RFT (MDT) data	258
9.6.3. PLT data	259
10. Waterflood	261
10.1. Stream Lines. Stream Line Settings	263
10.2. Well drainage zone	266
10.3. Drainage matrix, graph, table, network	269
10.3.1. Drainage Table	271
10.3.2. Drainage Graph	275
10.3.3. Drainage matrix	277
10.3.4. Drainage network	278
10.4. Balancing	280
10.5. Waterflood compensation	283

11. 2D Histogram. Crossplot	284
11.1. 2D Histogram	284
11.2. X/Y Histogram	288
11.3. Crossplot	291
12. Fluid Properties	293
12.1. Properties editing	295
12.2. Properties. Right Panel Buttons	297
12.3. SRP (Scaled Relative Permeability Parameters)	298
12.4. Hysteresis visualization	299
12.5. Rates vs. SWAT	300
12.6. Flow Function	301
12.7. Propants	303
12.8. Chemical Properties	306
12.9. VFP tables	307
13. Economical parameters	308
13.1. Setting Economics Parameters	308
13.2. Net Present Value Graph	309
14. User Arithmetic	313
14.1. Available User Maps and Operations	314
14.2. Scaling User Properties	319
14.3. Arithmetic operations	321
14.4. Difference in arithmetic usage in interface and in files	323
14.5. Examples	324
14.5.1. Unary and Binary operations	324
14.5.2. Logical operators	324
14.5.3. Local changes in internal areas of a property	325
14.5.4. Examples for user properties (maps)	326
14.5.5. Examples for user cuts	326
14.6. Functions for User Maps	327
14.6.1. Examples	328
14.7. Functions for Wells	330
14.7.1. Functions for Single Wells	331
14.7.2. Combining wells under common mask	332
14.7.3. Functions for wells	333
14.7.4. Functions for blocks	333
14.7.5. Filters for well data	334
15. Property editing. Smoothing. Interpolation	336
15.1. Calculator for User Cuts and User Maps	337
15.2. Region Brush	338
15.3. User Cuts	339

15.3.1. Load FIP Boundaries to User Cuts or to User Maps	344
15.3.2. Save a Report for the Well Side Tracks	346
15.3.3. How to Use a User Cut to Add an Aquifer	348
15.4. User Maps	349
15.4.1. How to Add the Created User Map to the Model	351
15.5. Property Editing	352
15.5.1. Arithmetics	354
15.5.2. Block	355
15.5.3. Cylinder	356
15.5.4. Wells	359
15.5.5. Profile	360
15.5.6. Cross-Section	363
15.5.7. Wells data	365
15.5.8. 3D Grid Properties Data	366
15.5.9. Stream lines	367
15.5.10. Derivative Maps	367
15.5.11. Voronoi Diagrams	367
15.5.12. Connected components	368
15.5.13. Faults	369
15.6. Smoothing	370
15.7. Interpolation methods	371
15.7.1. Least Squares method	371
15.7.2. Trivial interpolation method	373
15.7.3. Multilayer IDW method	374
15.7.4. Kriging	375
15.7.5. Sequential Gaussian Simulation (SGS) method	377
15.8. Interpolation	378
15.8.1. Interpolation by Multilayer Least Squares method	380
15.8.2. Interpolation by 3D Least Squares method	382
15.8.3. Interpolation by Multilayer Kriging	383
15.8.4. Interpolation by 3D Kriging	386
15.8.5. Interpolation by Multilayer SGS method	387
15.8.6. Interpolation by 3D SGS method	389
15.8.7. Interpolation by trivial interpolation method	390
15.8.8. Interpolation by multilayer IDW method	391
15.9. Permeability Multiplier	392
16. Field Development Planning	394
16.1. Adding a well. Forecast. Tracers	395
16.2. Hydraulic fracture	396
16.3. Well bottomhole zone treatment (BHZT)	398

17. tNavigator settings	399
17.1. tNavigator's General Settings	400
17.2. Models	402
17.3. Paths	404
17.4. Graphics	405
17.5. Maps and Graphs	407
17.6. Strings	408
17.7. Update settings	412
17.8. Client Options	413
17.9. Advanced	414
17.10. Designer	415
17.11. Preferences	416
17.11.1. Well And Connection Icons	417
18. References	420

1. Introduction

tNavigator is a software package, offered as a single executable, which allows to build static and dynamic reservoir models, run dynamic simulations, perform extended uncertainty analysis and build surface network as a part of one integrated workflow. All the parts of the workflow share common proprietary internal data storage system, super-scalable parallel numerical engine, data input/output mechanism and graphical user interface. tNavigator supports METRIC, LAB, FIELD units systems.

tNavigator is a multi-platform software application written in C++ and can be installed on Linux, Windows 64-bit OS and run on systems with shared and distributed memory layout as a console or GUI (local or remote) based application. tNavigator runs on workstations and clusters. Cloud based solution with full GUI capabilities via remote desktop is also available.

tNavigator contains the following 8 functional modules licensed separately:

- Geology Designer (includes PVT Designer and VFP Designer);
- Model Designer (includes PVT Designer and VFP Designer);
- Network Designer (includes PVT Designer and VFP Designer);
- Black Oil simulator;
- Compositional simulator;
- Thermal simulator;
- Assisted History Matching (AHM, optimization and uncertainty analysis);
- Graphical User Interface.

The list of tNavigator documentation is available in tNavigator Library.

In this document there is a description of **Graphical User Interface**, that is fully integrated with simulation modules (Black Oil simulator, Compositional simulator, Thermal simulator).

tNavigator User Manual contains the description of physical model, mathematical model and the keywords that can be used in dynamic model.

Graphical User Interface allows to edit dynamic model in a single graphical user interface, reflecting changes in 2D, 3D and graphs, and run model computations demonstrating the computation process. The user may modify the dynamic model during simulation interactively and review the results during or after the calculation (tNavigator presents the results as tables, graphs, bubble maps, 2D and 3D, well sections, crossplots and various forms of reports).

2. tNavigator documentation

Technical documentation for all tNavigator modules is available in Russian and English. The list is in the document tNavigator Library.

Documents can be opened from tNavigator's main window by choosing tab **Manuals** and pressing on **Manuals List** button (see figure 1). The technical descriptions are available in the language corresponding to the current language of the interface. Clicking on the button **Export** in the menu **Manuals** will lead to the export of all manuals. In addition any manual can be opened using menu **Help**.



Figure 1. Manuals

The newest version of tNavigator manuals for users (and all training tutorials with test models) is available on support site <https://support.rfdyn.com>. You will need to enter your login and password twice.

Links to the documentation are on the left bottom corner of the page (figure 2).

- **View docs in English** – view documentation in English.
- **View docs in Russian** – view documentation in Russian.

- **Download docs ENG** – download documentation in English (including all training courses and test models).
- **Download docs RUS** – download documentation in Russian (including all training courses and test models).



Rock Flow Dynamics © 2005-2017

Figure 2. Download the newest tNavigator's documentation

- **Create Forecast.**

This will pop up the dialogue **Creating Forecast Model**. See detailed description in the training course **How To Use Restart**.

- **Create History Matching Project.**

Choosing this option allows to create a History Matching Project. You can create the project using existing, already specified variables or you can set them using the **History Matching Variables Manager** or directly in the data file via the keyword **DEFINES** (see 12.1.21).

The detailed description of algorithms, the objective function calculation, the uncertainty analysis is in the User Guide for History Matching Module (document **tNavAHMUserGuideEnglish**).

- **History Matching Variables Manager.**

Allows to set the variables for AHM according to available scenarios.

See training courses:

8.1 AHM Theoretical Course,

8.2 How To Use Assisted History Matching,

8.3 How To Use RFT in History Matching,

8.4 How To Find The Best Well Trajectory,

8.5 Corey RP in AHM,

8.6 How To Use AHM for Hydraulic Fracture,

8.7 How To Run Different GEO Realizations.

- **Export Settings.** Allows to set properties and graph settings for one model only and then apply these settings to other models. Settings: graph colors, Graphs Templates, graphs in User selection, colors for properties, visualization options.

(a) Set required settings in one model. Press **Document. Export Settings**. All the settings will be saved in the .tNav-file.

(b) Open a new model (to which the created settings should be applied). **Document. Import Settings**.

(c) Select .tNav-file to be applied to the current model.

- **Import Settings.**

Import file with settings of graphs style, Graph Templates, graphs in User Selection, properties colors and visualisation options. See paragraph **Export Settings**.

- **Load Well Data.**

Load well data from text files: well trajectories, layers, events, history, well logs, RFT (MDT), PLT.

- **Load results.**

Graphs of different runs can be compared in one window (for wells, groups, etc.) – 7.10.

Computation data of tNavigator, Eclipse or MORE can be loaded. See the training

course 1.5 How To Load Maps And Graphs.

This option can be used to compare various versions of the field development forecast. All the graphs will show graphs corresponding to the added model. The added models' results will be superimposed on the initial model data. If reporting steps are not the same, the results will be interpolated into the initial model's steps; If wells from loaded model are not existed in the initial model they will be ignored, etc.

Grid properties can be loaded here in the following formats:

- **GRD file[M] – .grd**

File type: binary file, generated via Roxar MORE.

File format – .grd.

Data description: you will be offered a choice of which cubes of properties available in the file to load.

- **Array file[M] – .ara**

File type: binary file, generated via Roxar MORE.

File format – .ara.

Data description: you will be offered a choice of which cubes of properties available in the file to load and for which time steps.

- **Restart file – .UNRST**

File type: binary file, generated via Eclipse.

File format – .UNRST.

Data description: you will be offered a choice of which cubes of properties available in the file to load and for which time steps.

- **Show Loaded Results.**

Show loaded models' results.

- **Preferences.**

There you can configure Visualization, Well options, Contour lines, Streamlines, Drainage network;

- **Economics Preferences.**

See the detailed description of economic parameters and Net Present Value graph in the section Economics Preferences.

- **Calculation Parameters.**

This item allows to see and edit parameters of the iteration process (the keyword **RUNCTRL**, see 12.18.124).

- **Close.**

Close the model.

Hot key – **Ctrl+Q**.

2. Menu View.

- **New.** Create New View. Create an additional window for the current model. It is possible to create several windows for the same model for simultaneous viewing graphs and properties in different windows.
Create Quick Graph View. Hot key – **Ctrl+N**. This will create a new window with graphs for the current model.
- **Show all.**
Show all windows created for the current model.
- **Hide all.**
Hide all windows created for the current model.
- **Close.**
Close all the windows additionally created for the current model (except for the model's main window).

3. Menu Files.

A full list of the current model's files. Clicking on a file will open it in a text editor. The text editor to view files can be set via Settings. Options. Path in tNavigator main window.

4. Reports.

See the training tutorial **1.3 How To Import Export Data Reports.**

3.7. (Status) Report Panel – log

In the bottom part of the window, there is a panel showing the loading and computation status of the model. It shows brief information about the model's loading status and for each time step. General information is shown in green, errors are shown in red, and warnings are shown in orange. The level of detail is driven by the settings in .data file defined by the keywords REPORTFILE (see 12.1.3) and REPORTSCREEN (see 12.1.3). These keywords are setting references of log panel.

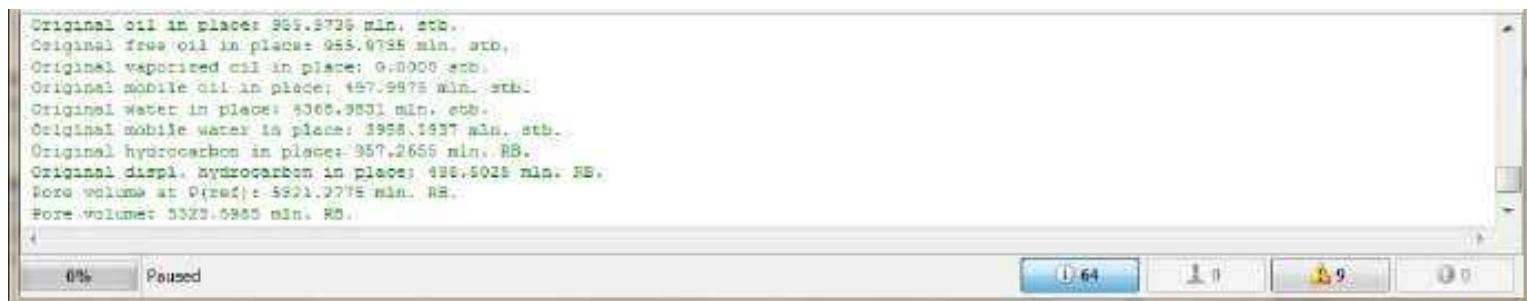


Figure 6. Report Panel.

The default setting is to show all messages (button **Messages** on the bottom panel). To have warnings and errors only shown, click button **Warnings and Errors** on the bottom panel.

Elapsed shows the time passed from the begining of the computation. **Estimated** shows the time left to the completion of the computation.

A full report about tNavigator's work with the model can also be viewed in the file **MODEL_TITLE.log** in the **RESULTS** folder (the folder is created near the model's .data file).

Search in log.

To use **Search** in the log panel press one mouse click on the panel and press **Ctrl+F**. Enter the text in the appeared row – figure 7.

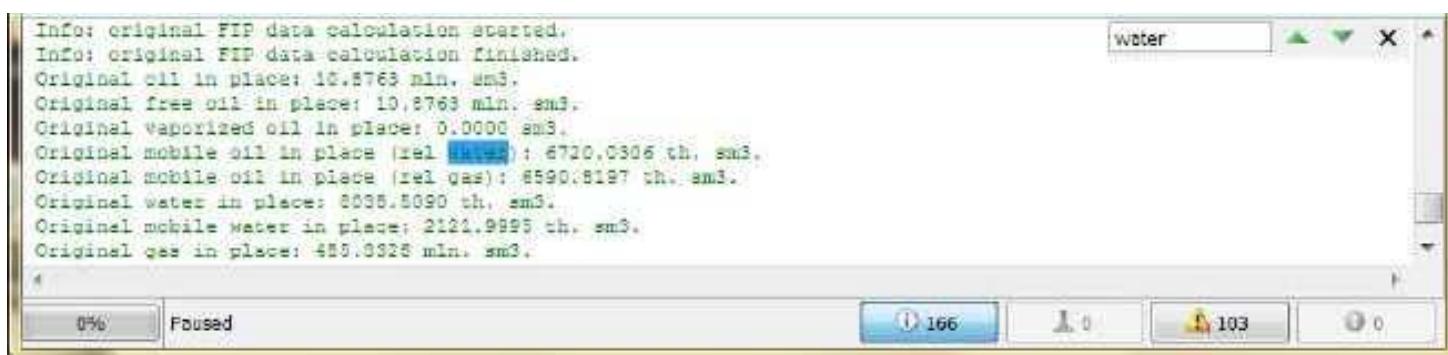


Figure 7. Search in the log.

Double click on any warning or error shown in the log-panel will open the file to which this warning or error refers (see figure 8). The line of the file which contains error will be highlighted.

In the dialog **Schedule Options** you can set SCHEDULE section saving format. It is available only if the option **Export Schedule Data** or **Custom** was selected. The dialog contains the following options:

- **Default.** SCHEDULE will be exported in tNavigator format.
- **E100/E300 Format.** SCHEDULE will be exported in E100/E300 format. In this case keywords which can be used only for tNavigator will not be exported; you will see a warning about it at the bottom of the dialog window.
- **Save Perforations in E100/E300 Format.**
- **Keep Schedule Syntax.** Export SCHEDULE section in the initial format. The option is available only for the hybrid or MORE type models.

When saving the file specify a full path and the name of the file.

3.8.5. Split model



This feature can only be used before running a model computation. See the detailed instructions on the model splitting in the section Split and merge (sector modeling) of **tNavUserManual**.

See the training tutorial [7.1 How To Split And Merge Model](#).

3.8.6. History Matching Variables Manager



Create History Matching Project – The window to create a History Matching Project will be opened. You can create a project using existing, already specified variables or you can set them in the **History Matching Variables Manager** or directly in the data file via the keyword **DEFINES** (see 12.1.21).

The detailed description of all algorithms, the objective function calculation, the uncertainty analysis is in the User Guide for History Matching Module (document **tNavAHMUser-GuideEnglish**). **History Matching Variables Manager** – Allows to set the variables for AHM according to available scenarios.

See the training courses:

- 8.1 AHM Theoretical Course,**
- 8.2 How To Use Assisted History Matching,**
- 8.3 How To Use RFT in History Matching,**
- 8.4 How To Find The Best Well Trajectory,**
- 8.5 Corey RP in AHM,**
- 8.6 How To Use AHM for Hydraulic Fracture,**
- 8.7 How To Run Different GEO Realizations.**

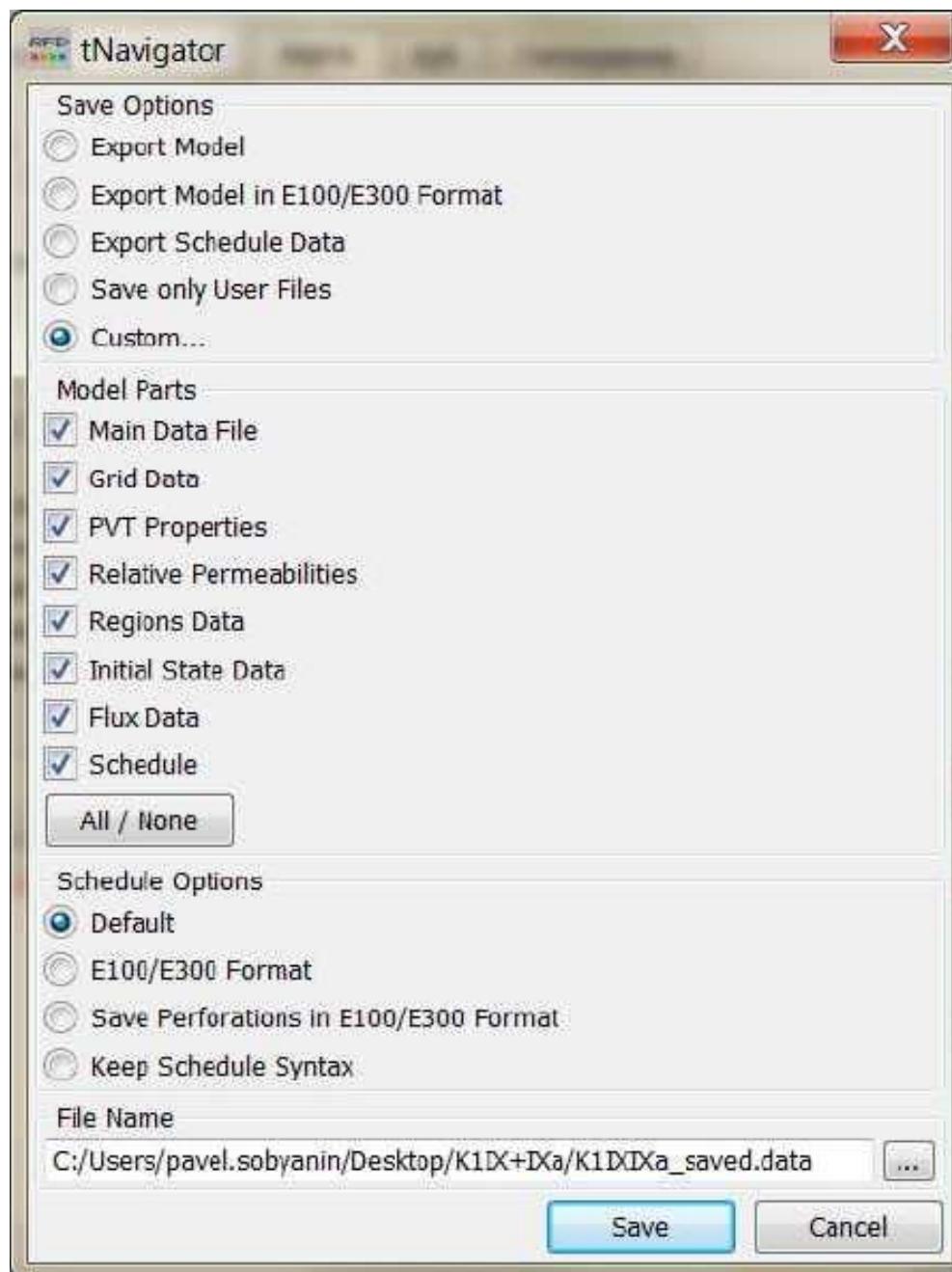


Figure 11. To save a model dialog.

3.8.7. Reload Model

 – **Reload Model.**

See the detailed description in the section Document. View. Files. Reports

3.8.8. Run calculation

 – **Run.**

Runs model calculations. Calculation details. You can run a calculations by pressing **Ctrl+R**. The time slider will move by time step. The steps completed will be redlined.

3.8.9. Playback Results



– Playback.

Auto playback calculated time step results (step changes in properties, graphs, and tables).

3.8.10. Stop



– Stop.

Stop (pause) calculations. You can resume a computation where you stopped it or from any previously computed time step.

3.9. Left panel buttons

Description of left panel buttons.

-  **Show list of visualization methods and tree of corresponding objects.**

Press this button to show the options panel and tree of corresponding objects. Second pressing on this button hides this panel.

-  **Show visualization properties.**

Press this button to show the panel of current object visualization settings. Second pressing on this button hides this panel.

3.10. Run model

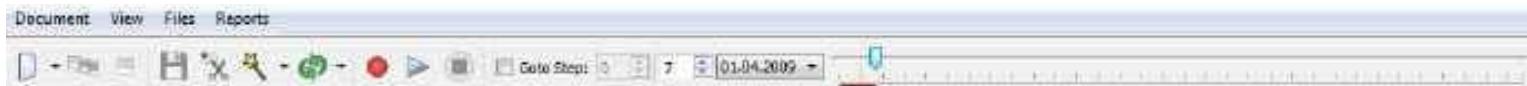


Figure 12. Run and stop calculation panel

To run calculations from step 0, click **Run** regardless of the position of the time step slider. Calculations can be run from any previously calculated time step. You can run calculations by pressing **Ctrl+R**.

To stop computations at any step, click **Stop**. Putting the mouse pointer on the time slider will display the date of the time step corresponding to the slider's current position.

3.10.1. Compute to a fixed step

To run calculations from step 0 to a fixed step:

1. Check **Goto Step** on the top panel;
2. Select the time step number in the neighboring field;
3. Click **Run**.

The calculation will stop when the selected time step is reached.

3.10.2. Re-Run Calculations

You can re-run a calculation from any step:

1. Pause the calculation by clicking **Stop**.
2. Wait for the calculation to stop (for the processor loading indicator to stop rotating).
3. Use the left mouse button to move the time slider to the step from which you want the computation to resume.
4. Click **Run**.
5. You may re-Run calculations as many times as you need.

Remark. The time slider cannot be moved during a calculation (if **Stop** button is not pressed). The slider shows the number of the current time step. The slider can only be moved after the calculation is stopped.

3.10.3. Start Calculations from Any Previously Calculated Step

This feature can be used as follows: run calculations for any time step number, close the model, re-open the model's data file (ModelName.data), set the time slider at any previously calculated time step and press **Run**. Previously calculated steps are marked with a red line under the time slider.

3.11. tNavigator hotkeys

The following hotkeys are available in tNavigator:

- **Ctrl+R** – run calculations.
- **Ctrl+P** – make a screenshot.
- **Ctrl+S** – save model.
- **Ctrl+Q** – close model.
- **Ctrl+N** – create Quick Graph View.
- **Ctrl+L** – reload model.
- **Ctrl+Shift+L** – clear results and reload model (all files in the RESULTS folder will be deleted, and a model will be re-opened).
- **Ctrl+click** (on property) – open dialog **Well properties** for editing the nearest well to click.
- **Alt+click** (on property) – add new well (producing, injecting or well template – depending on settings).
- **Double click on well** (on property) – jump to Rates graphs for this well.
- **Double click on block** (on property) – jump to Block info graph for this block.
- **Simultaneous clicking left and right mouse buttons** or **Ctrl+0** (for 2D, 3D and Graphs) – default view.

3.12. Definitions

Definitions contains general information about model (figure 13):

- Model title;
- Starting date;
- Language;
- Model type;
- Collector type (Single porosity, Dual porosity);
- Dimensions NX, NY, NZ (the number of blocks which model divided along X, Y, Z axes);
- Total block number, active block number;
- Wells number, transit wells, well groups, the number of connections, maximal connections number per well;
- Information about polymer, alkaline, surfactant, brine, tracers data in the model (On / Off).

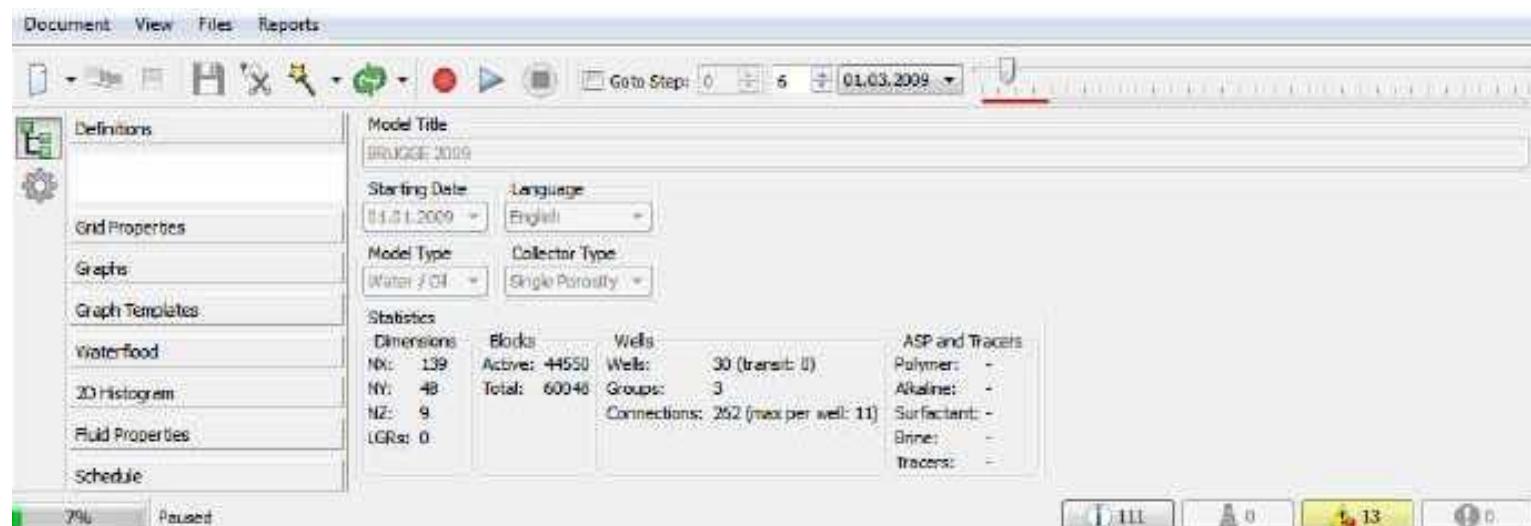


Figure 13. Definitions

3.13. Schedule

The option displays a well geometry and well production parameters set in a model. All the data are displayed in the tabular form. The tables can be sorted as in the **Graphs** option. Tables display the parameters, specified by keywords in the initial model, regarding wells. Move the mouse on the keyword or its parameter to see the pop-up tip.

The **Schedule** (Well Data) option displays the following tabs:

- **Well Definitions.** The data in the table correspond to the keywords **WELSPECS** (see 12.18.3), **COMPDAT** (see 12.18.6), **WPIMULT** (see 12.18.30), **COMPFRAC** (see 12.18.131), **WFRAC** (see 12.18.127). The table presents the following well data: commissioning date and perforation jobs. The columns are: Date, Operation, Well Name, Group Name, Connection Blocks (I, J, k1, k2), Status, Diameter, Skin, Direction, Productivity Index Multiplier and Fracture Azimuth Angle.

Date	Operation	Description	Well Name	Group Name	Operation Parameters
01.10.2004	WELSPECS	Well specification	WELL8	"	WELL8" 11 9 1650 OIL *** YES
01.10.2004	COMPORD	Order of well connections	WELL8	"	WELL8 LASTINPT
01.10.2004	COMPDAT	Perforation	WELL8	"	WELL8 11 9 2 2 OPEN * 0.120468 0.216 11.3276 *** 19.6275
01.10.2004	COMPDAT	Perforation	WELL8	"	WELL8 11 9 4 4 OPEN * 0.085304 0.216 8.15054 *** 19.6515
01.10.2004	COMPDAT	Perforation	WELL8	"	WELL8 11 9 6 6 OPEN * 0.0625246 0.216 5.90597 *** 19.7133
01.10.2004	COMPDAT	Perforation	WELL8	"	WELL8 11 9 7 7 OPEN * 0.0472558 0.216 4.40981 *** 19.7133
01.10.2004	COMPDAT	Perforation	WELL8	"	WELL8 11 9 8 8 OPEN * 0.128322 0.216 12.192 *** 19.6953
01.10.2004	COMPDAT	Perforation	WELL8	"	WELL8 11 9 9 9 OPEN * 0.00430372 0.216 0.366196 *** 19.6857
01.10.2004	COMPDAT	Perforation	WELL8	"	WELL8 11 9 10 10 OPEN * 0.755373 0.216 70.8424 *** 19.6775
01.10.2004	COMPDAT	Perforation	WELL8	"	WELL8 11 9 11 11 OPEN * 4.25587 0.216 404.587 *** 19.6775
01.10.2004	COMPDAT	Perforation	WELL8	"	WELL8 11 9 12 12 OPEN * 0.310641 0.216 29.0575 *** 19.6775
01.10.2004	COMPDAT	Perforation	WELL8	"	WELL8 11 9 14 14 SHUT ** 0.216
01.10.2004	COMPDAT	Perforation	WELL8	"	WELL8 11 9 22 23 SHUT ** 0.216
01.03.2005	WELSPECS	Well specification	WELL10	"	WELL10" 7 5 1650 OIL *** YES
01.03.2005	COMPORD	Order of well connections	WELL10	"	WELL10 LASTINPT
01.03.2005	COMPDAT	Perforation	WELL10	"	WELL10 7 5 11 11 OPEN * 0.106801 0.216 10.1123 *** 19.6859
01.03.2005	COMPDAT	Perforation	WELL10	"	WELL10 7 5 17 17 OPEN * 5.16222 0.216 471.581 *** 19.6859
01.03.2005	COMPDAT	Perforation	WELL10	"	WELL10 7 6 13 13 OPEN * 9.7484 0.216 531.567 *** 19.6933
01.03.2005	COMPDAT	Perforation	WELL10	"	WELL10 7 6 14 14 OPEN * 3.67044 0.216 350.281 *** 19.7011
01.03.2005	COMPDAT	Perforation	WELL10	"	WELL10 7 6 15 15 OPEN * 1 transmissibility factor for the connection
01.03.2005	COMPDAT	Perforation	WELL10	"	WELL10 7 6 16 16 OPEN * 1.06355 0.216 101.183 *** 19.705

Figure 14. Schedule. Well definition.

- **Well Production.** The data in the table correspond to the keywords **WCONHIST** (see 12.18.37), **WCONPROD** (see 12.18.36), **WCONINJE** (see 12.18.38), etc. The table presents production history data and specifies production rate and pressure caps. The columns are: Date, Operation, Well Name, Status, etc.

• Multisegment wells.

The table presents the following well data: **COMPSEGL** (see 12.18.23), **COMPSEGS** (see 12.18.22), **WELSEGS** (see 12.18.11), **WSEGAIID** (see 12.18.15), **WSEGEXSS** (see 12.18.17), **WSEGFLIM** (see 12.18.18), **WSEGITER** (see 12.18.118), **WSEGTABLE** (see 12.18.13), **WSEGVALV** (see 12.18.14) and other.

4.4. Regions

Regions are shown in different colors.

Designation (tNavigator keyword)	Map Description
PVT Regions (PVTNUM, see 12.4.2)	PVT property regions. Regions have the following parameters assigned: reference pressure, formation volume pressure, compressibility factor, viscosity.
Saturation function Regions (SATNUM, see 12.4.3)	Filtration regions. For each region, relative permeabilities to saturation ratios are assigned.
IMBNUM (see 12.4.7) (filtration regions at imbibition)	It is used to specify hysteresis of relative permeabilities – Hysteresis (option HYSTER in the keyword SATOPTS, see 12.1.71). This keyword specifies which saturation tables should be used for each block during imbibition. For processes of drainage and equilibrium the keyword SATNUM (see 12.4.3) is used.
Rock Properties Regions (ROCKNUM, see 12.4.14)	For each rock properties region, a table of rock transmissibility vs. pressure is assigned.
Equilibrium Regions (EQLNUM, see 12.4.9)	For each equilibrium region, the model assigns parameters used for calculating initial conditions (the depth, the pressure at the depth, oil-water contact level, capillary pressure at the oil-water contact level, the gas-oil contact level, and the capillary pressure at the gas-oil contact level). All the blocks of an equilibrium region belong to the same PVT region and the same Filtration region.
SURFWNUM (see 12.4.5) (filtration regions for water-wettability case)	It is used to modelling ASP flooding, adsorption and influence on RP. In this case SATNUM sets RP regions of oil-wettability case, and SURFWNUM – for water-wettability case.
Active Blocks (ACTNUM, see 12.2.29)	Shows all active blocks.

FIPNUM (see 12.4.10) (Fluid-in-place Regions)	<p>All the reporting data will be shown for these regions. It is possible to view the following properties for fluid-in-place regions:</p> <ul style="list-style-type: none"> • The regions (properties showing the regions); • Water, oil, gas, and fluid rates, (H), under reservoir conditions; • Water, oil, and fluid production per month; • Water and gas injection rates, (H), under reservoir conditions; • Water, oil, or gas flow through the region's boundary under standard and reservoir conditions; • Aquifer water inflow. <p>Select a property in a pop-down menu at the bottom of the sub-options window – figure 19.</p>
FIPPATT (see 12.4.13)	Parts of original property after splitting

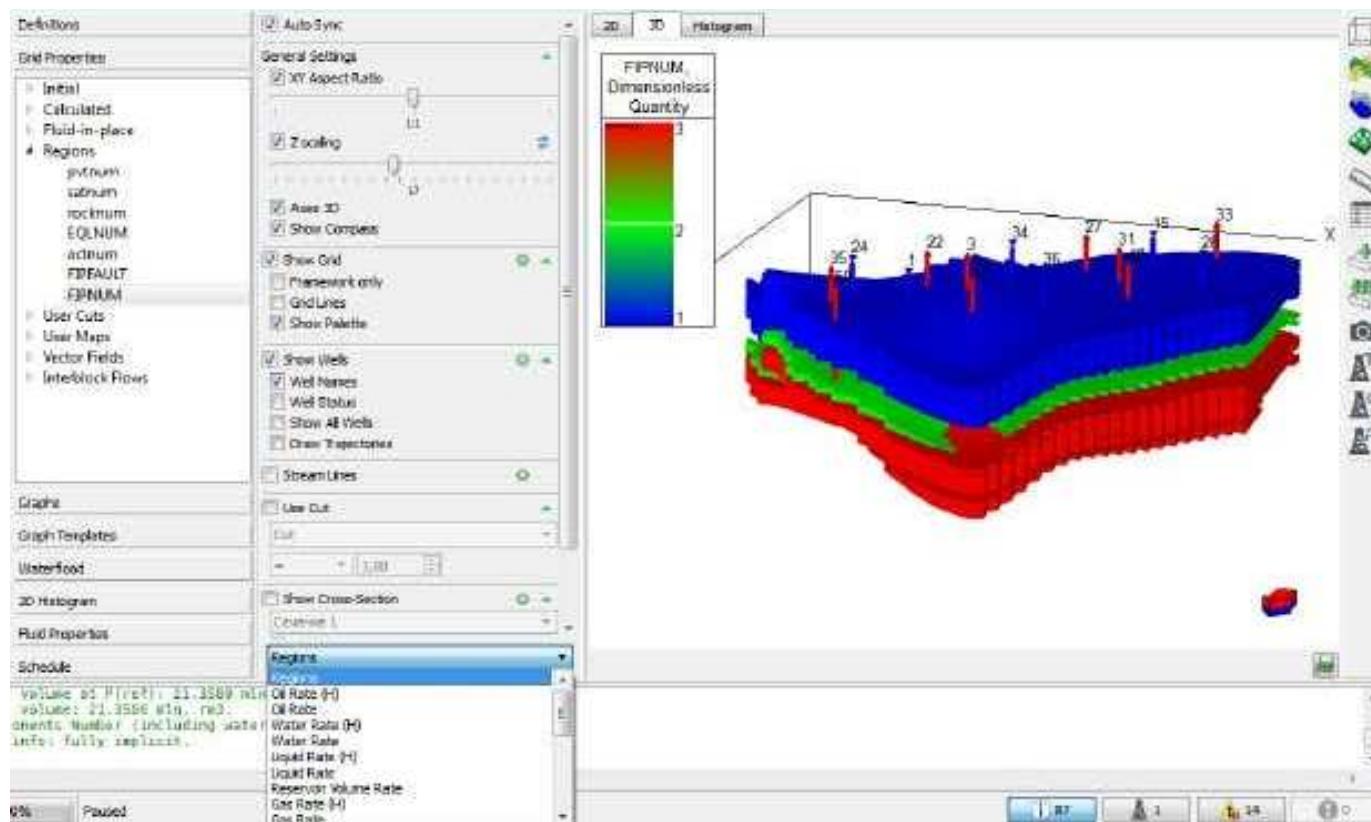


Figure 19. Menu of properties for FIP regions.

4.5. User Cuts, User Maps

Cuts

Cut	User Filters. You can create any number of User Filters: Cut, Cut1, Cut2, Cut3... See the detailed description in the section User Cuts.
-----	--

Maps

Map	User Maps. You can create any number of User Maps: Map, Map1, Map2, Map3... See the detailed description in the section User Maps.
-----	--

4.6. Vector Fields

Initial Stress	Initial stress value in each block – figure 20. These properties are available if the Geomechanics option is activated (keywords GEOMECH (see 12.1.94), ROCKAXES (see 12.5.21), ROCKSTRE , see 12.5.22)
Property of Stress Matrix Diagonal Elements	The magnitude of the vector comprising the diagonal elements of each block's stress tensor. These properties are available if the Geomechanics option is activated (keywords GEOMECH (see 12.1.94), ROCKAXES (see 12.5.21), ROCKSTRE , see 12.5.22). This property evaluates mesh block deformation in each block. These are matrix values, so the distribution shows a vector magnitude comprising diagonal elements. The diagonal element sign is shown as a unit vector whose components are normalized components of a vector comprising diagonal elements. The vector direction indicates axial compression (the corresponding vector component is less than 0) or axial expansion (the corresponding vector component is greater than 0). To display vectors, check Show Vector Field in the graphic interface. Vector size is controlled by a slider.

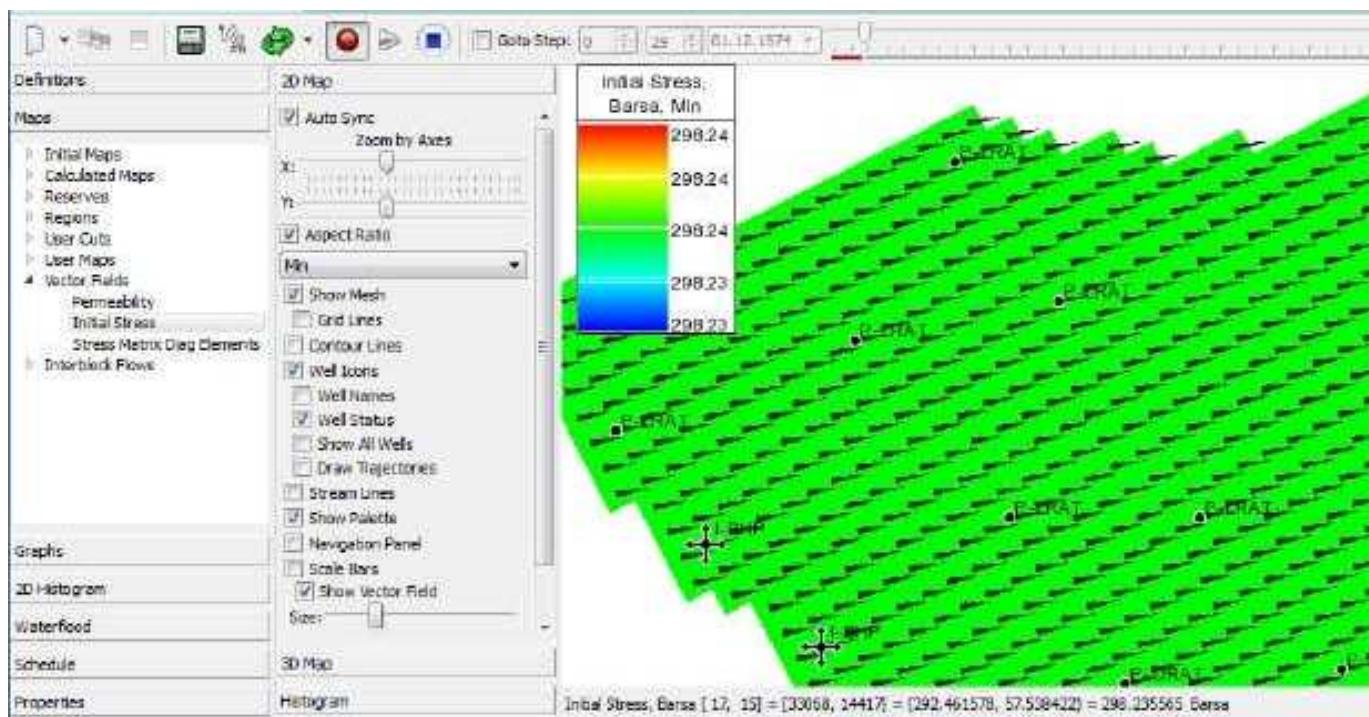


Figure 20. Initial Stress Map.

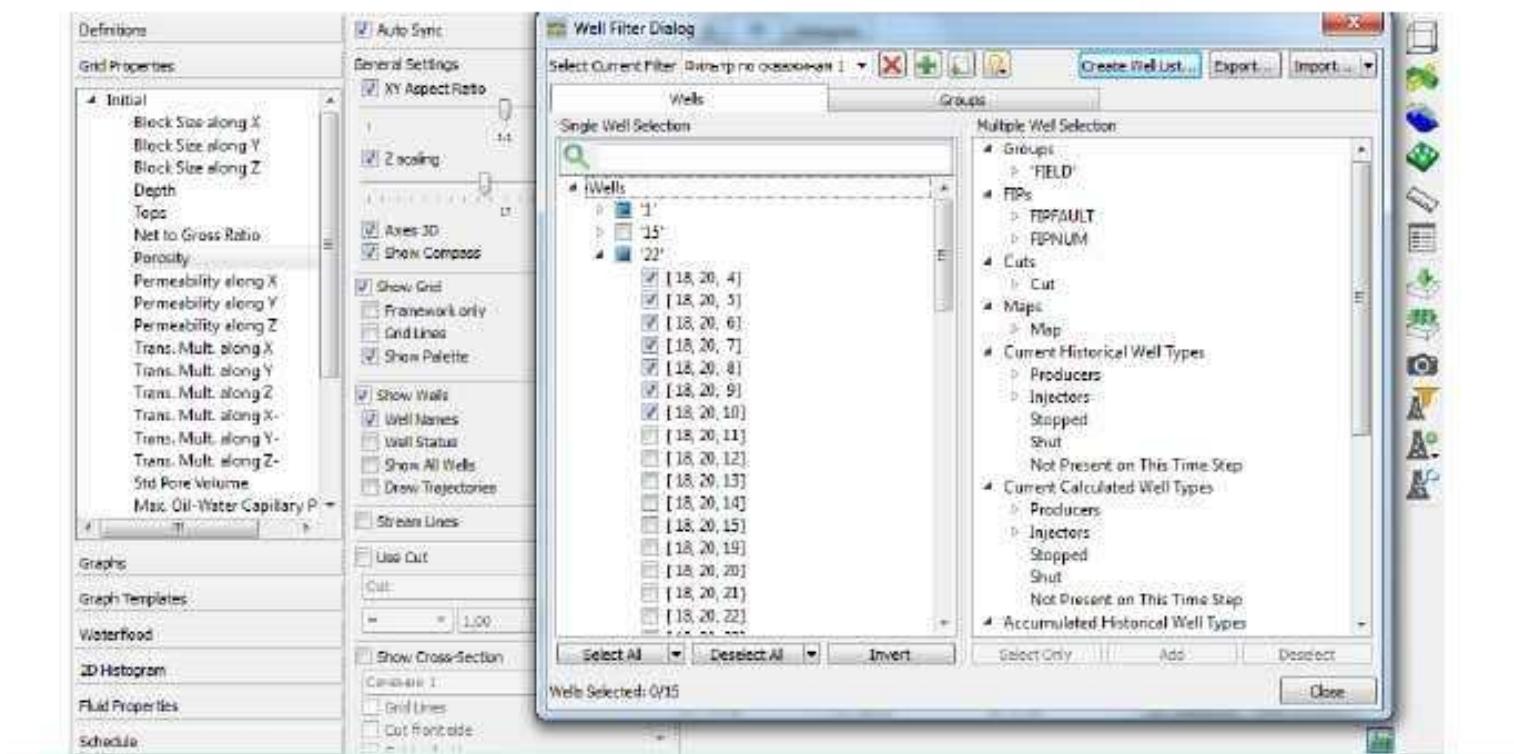


Figure 27. Well Filter created using a Slice Filter.

5.6. Create a Cross-Section

Create a Cross-Section.

This option allows to create vertical or horizontal cross-sections of the model. You can also create a cross-section via selected points or wells or well trajectories. The created cross-section can be viewed in 2D.

The difference between creating a Model Profile and creating a Model Cross-Section.

 [Create a Profile](#) and  [Create a Cross-Section](#) buttons. A profile automatically connects the centers of selected blocks by a **broken line** passing consecutively through the centers of neighboring blocks. To build the **straight-line** slice of the model (a depth section, a cross-section via the selected points, a well section, a well trajectory section), use the button

Create a Cross-Section.

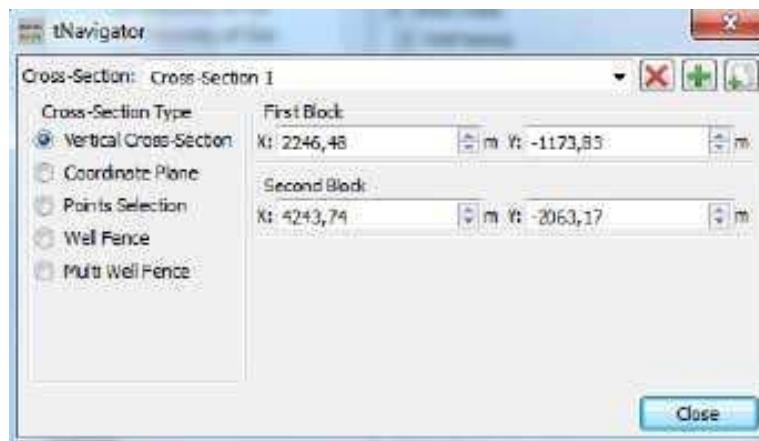


Figure 28. Create a Cross-Section/Fence.

There are several cross-sections types:

- Vertical Cross-Section;
- Coordinate Plane;
- Points Selection;
- Well Fence;
- Multi Well Fence.

The detailed description of creation of cross-sections is shown below.

[Creating a Vertical Cross-Section](#) – figure 29.

1. In the **Cross-Section** dialogue window select **Vertical Cross-Section**. Select start point and end point of the section by left-clicking on the model.
2. The appeared plane of the section can be moved by pulling the yellow balls. You can rotate the plane by pulling the corner cubics.

3. In the dialogue, you can manually set the cross-section's coordinates (in (METRIC: *m*, FIELD: *ft*)).
4. **Apply. OK.**

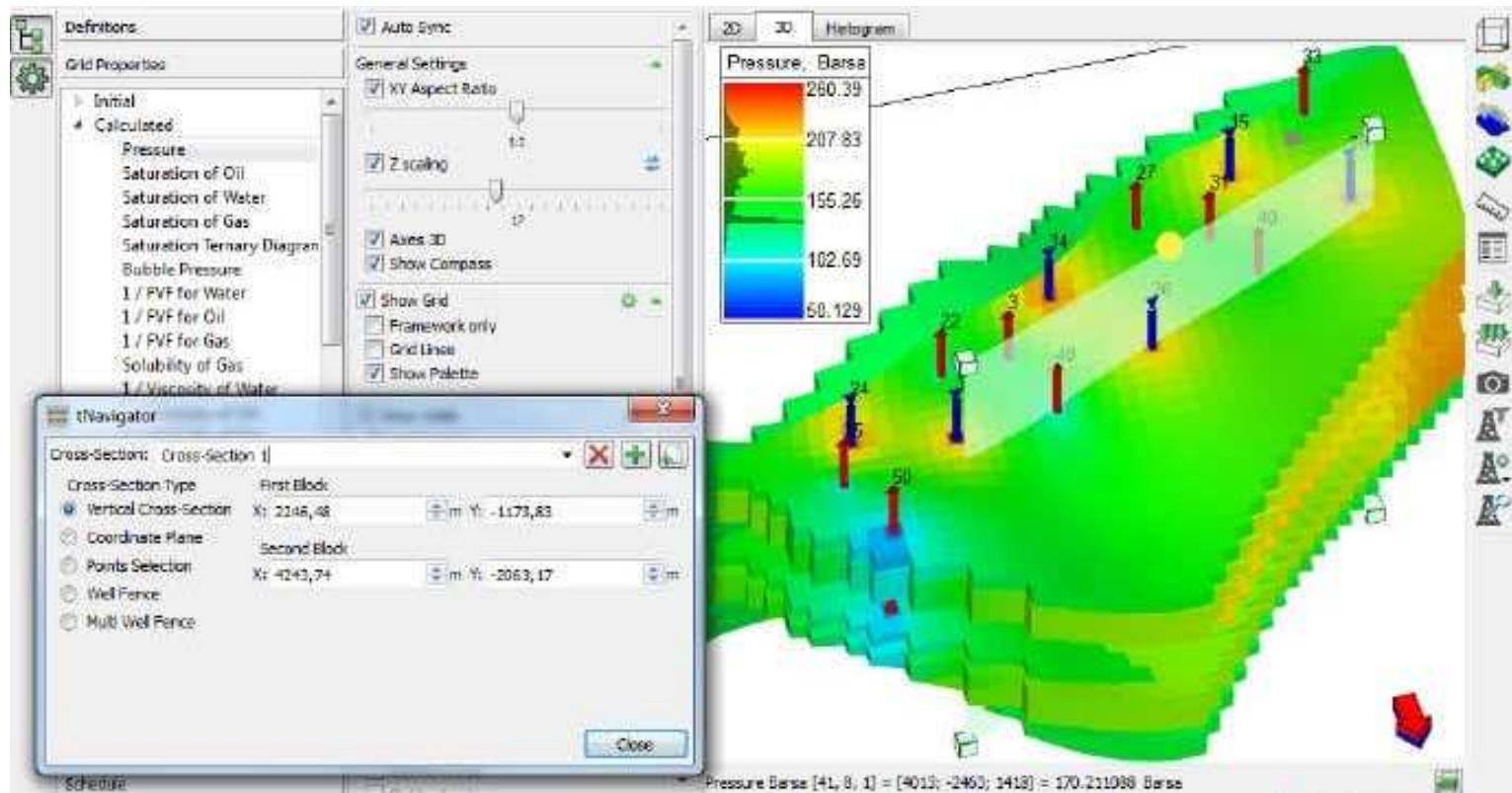


Figure 29. Create Vertical Cross-Section 1.

Creating a Coordinate Plane Cross-Section – figure 30.

1. In the **Cross-Section** dialogue window select **Coordinate Plane** and **Plane parallel to** which the cross-section will be created, e.g. Z plane.
2. Press the green plus button to **Add new cross-section**– Cross-Section 2.
3. The appeared plane of the section can be moved by pulling the yellow balls. You can rotate the plane by pulling the corner cubics.
4. If the cross-section crosses the boundary between blocks parallel to the Z plane you can select the blocks above or below the cross-section which faces will be located at Cross-Section.
5. In the dialogue, you can manually set the cross-section's depth (in (METRIC: *m*, FIELD: *ft*)).
6. **Apply. OK.**

Viewing Cross-Sections in 2D.

1. Go to a 2D. In the drop-down menu placed on the visualization settings tab, select **Cross-Section**.

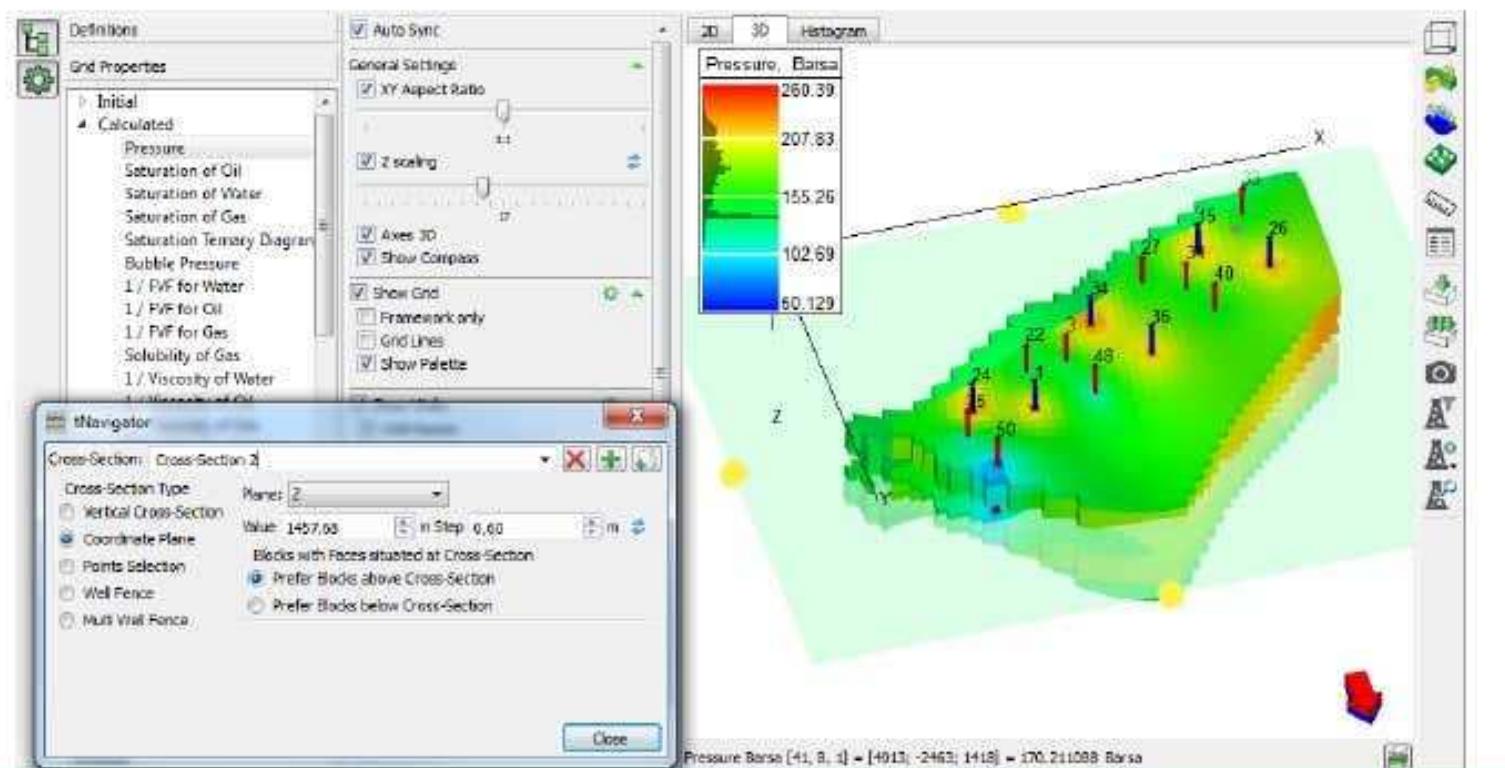


Figure 30. Create Coordinate Plane Cross-Section 2.

2. Cross-Section 1 is the created vertical cross-section – figure 31. Cross-Section 2 is the horizontal cross-section.
3. When viewing a vertical cross-section, it is recommended to uncheck **Aspect Ratio** (vertical sizes of blocks are small and difficult to analyze).

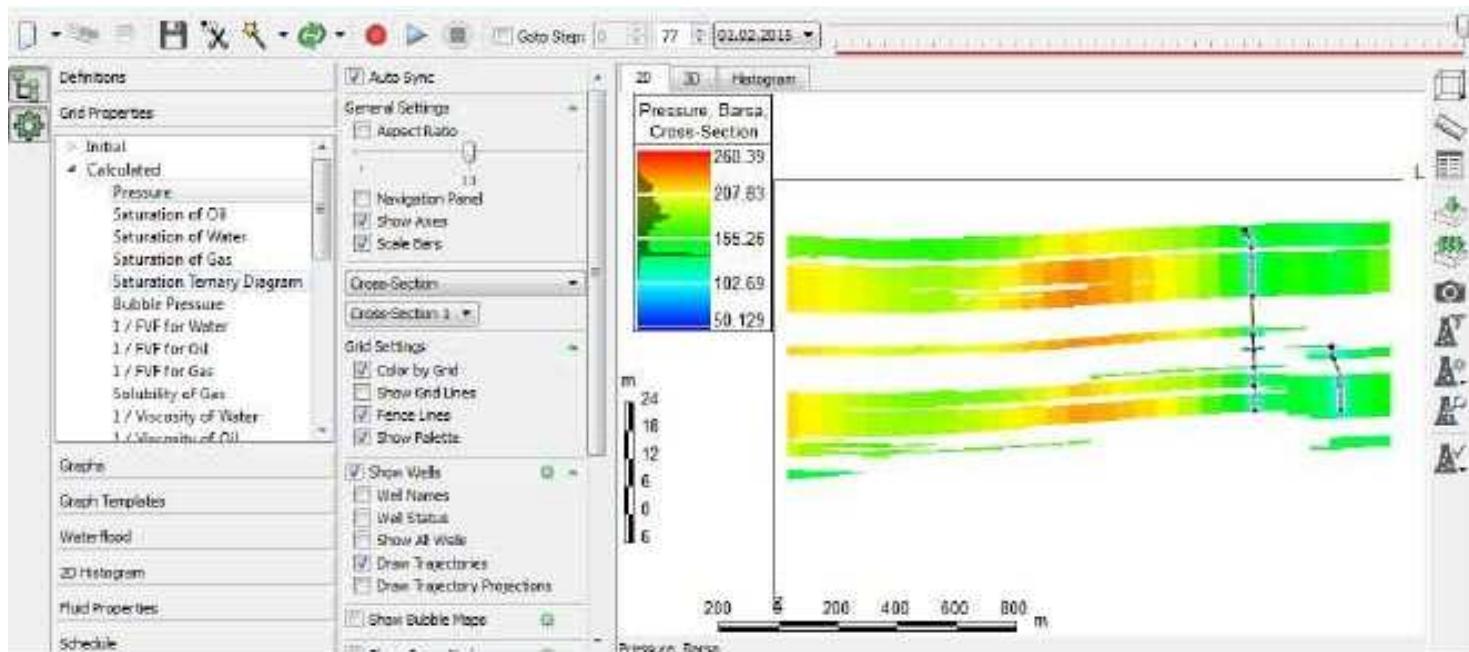


Figure 31. Cross-Section 1 is a vertical section.

Creating a Well Fence via a selection of points.

1. In the **Cross-Sections** dialogue select **Points Selection**.

2. Cross-Section. Add Cross-Section (pressing on the green plus)– Cross-Section 3.
3. Click the property to select the points for the construction of cross-section – figure32.
4. Press **Undo** to delete the last point.
5. The appeared fence can be moved by pulling the yellow balls.
6. You can change the position of fence points by pulling the cubic.

7. Apply. OK.

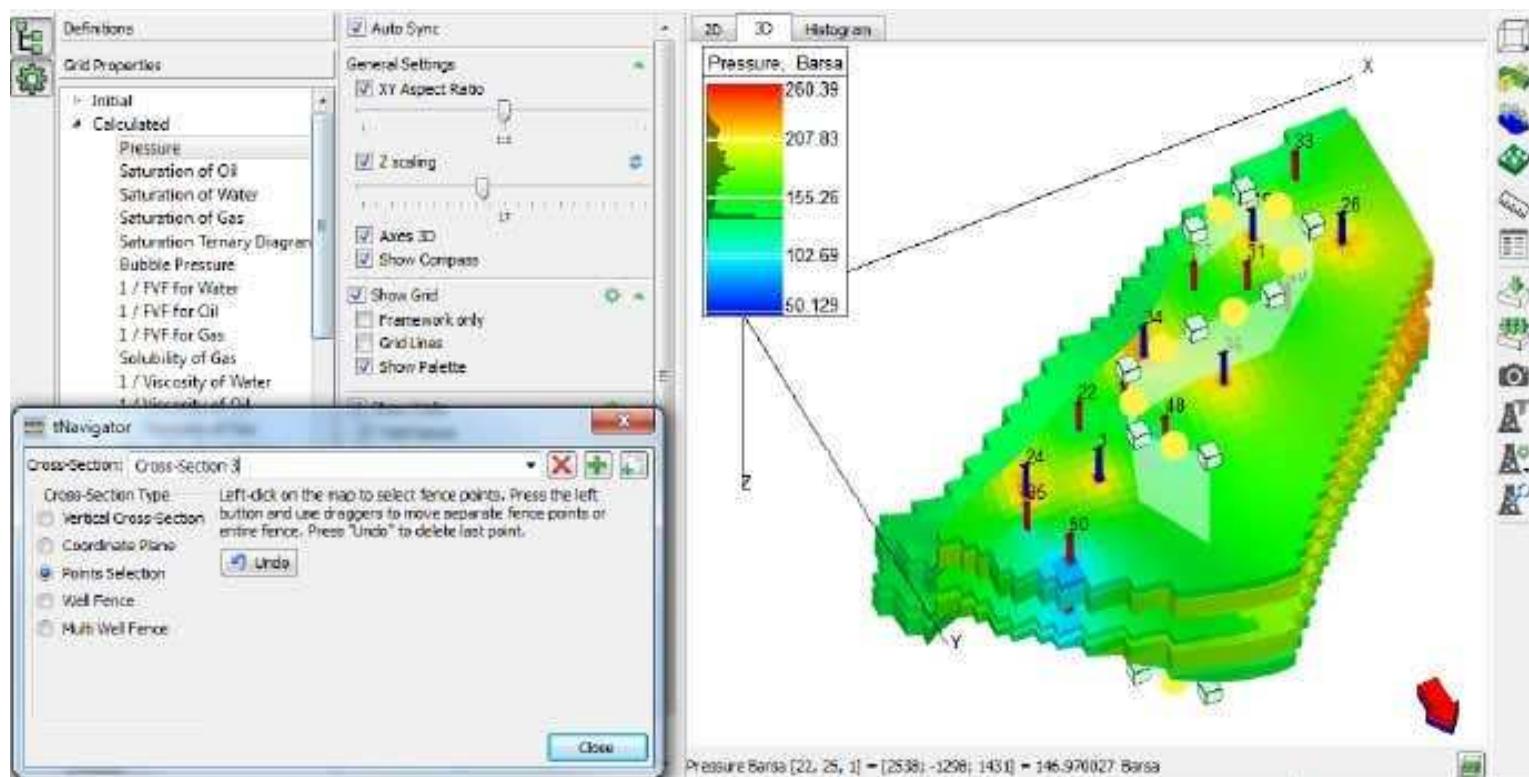


Figure 32. A points fence.

To view a cross-section:

8. Go to the 2D view. In the drop-down menu placed on the visualization settings tab select **Cross-Section–Cross-Section 3**.
9. Check **Fence Lines**. If this option is unchecked the vertical lines, corresponding to the fence points of cross-section, will be hidden.

Creating a Well Fence.

1. In the **Cross-Section** dialogue select **Well Fence**.
2. Cross-Section. Add Cross-Section (pressing on the green plus)– Cross-Section 4.
3. Select a well from the list. The selected well should be directional. Otherwise, if well is vertical this type of cross-section will be degenerate and will not be shown; figure 33.

4. If the well's trajectory has been loaded, you can check **Trajectory** and select the trajectory branch along which the fence will be created.
5. The **Tolerance** slider define a number of points of well's trajectory which will be used to create a well fence. The tolerance determines how far from the created line (defined the cross-section) skiped points of trajectory can be located.

Based on the slider position, the maximum distance (tolerance in the formula below), which should not be exceeded, is calculated according to the following formula:

$$\text{tolerance} = \frac{0.002 \cdot (1.0 - \text{slider_value}) \cdot \text{length}}{\text{n_points}},$$

where:

slider_value is the slider value from 0 to 1, **length** is the trajectory length, **n_points** is the number of trajectory points.

6. Apply. OK.

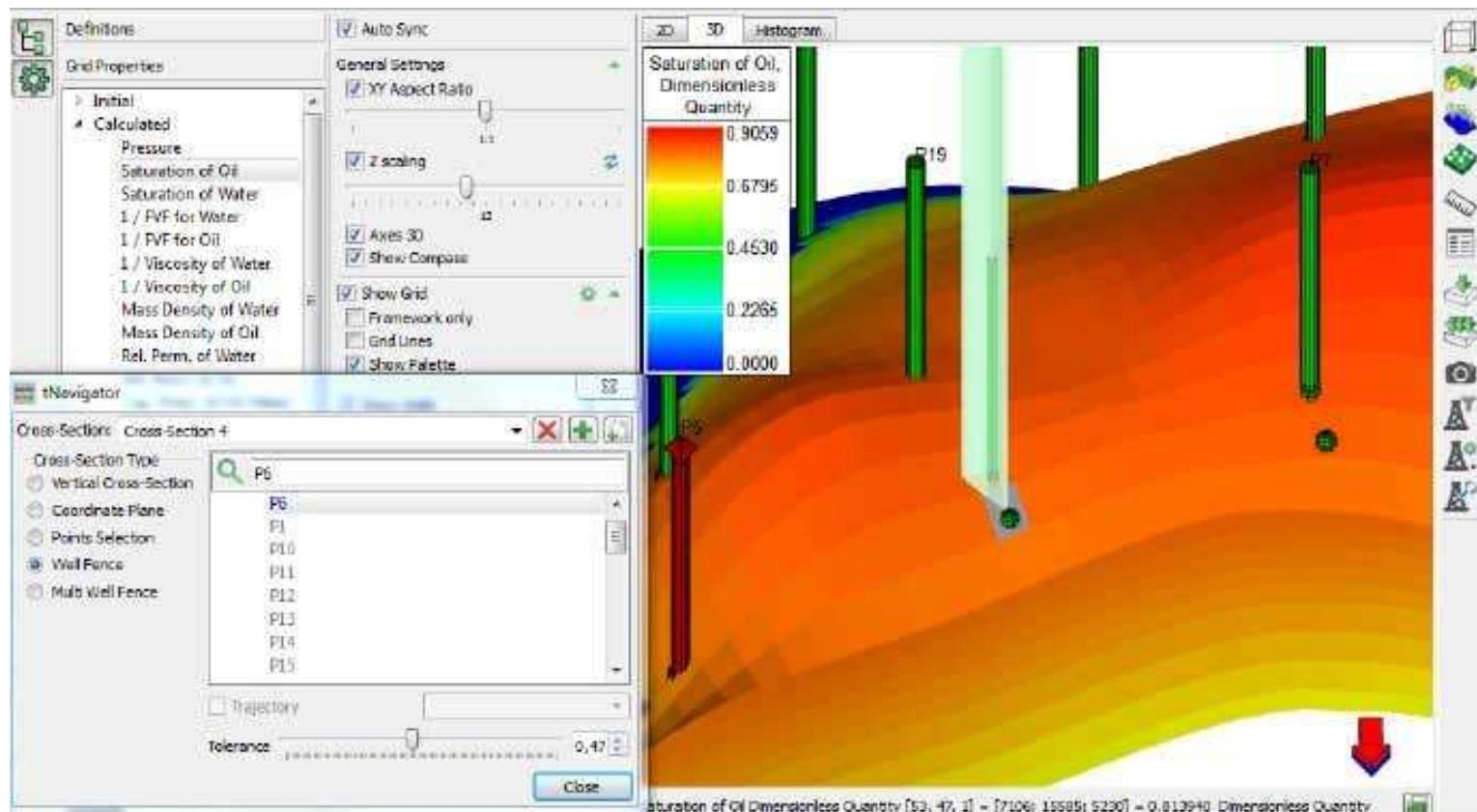


Figure 33. A well fence.

7. Go to the 2D view. In the drop-down menu placed on the visualization settings tab select **Cross-Section–Cross-Section 4**.
8. **Fence Lines** is checked (figure 33). If this option is unchecked the vertical lines, corresponding to the fence points of cross-section, will be hidden.

Creating a Multi-Well Fence.

1. In the **Cross-Sections** dialogue select **Multi-Well Fence**.
2. Cross-Section. Add Cross-Section (pressing on the green plus)– Cross-Section 5.
3. Select the wells based on which the fence will be created (you can select wells from **Well Filters** or from **Well Selection** – figure 34) in the drop-down menu.
4. If well trajectories have been loaded, you can check **Prefer Trajectories**.
5. **Close Fence** creates a closed fence (the first point and the last point will be connected).
6. **Alignment: Top, Middle, or Bottom** (for horizontal wells the fence line depends on the type of the alignment)
7. **Apply. OK.**

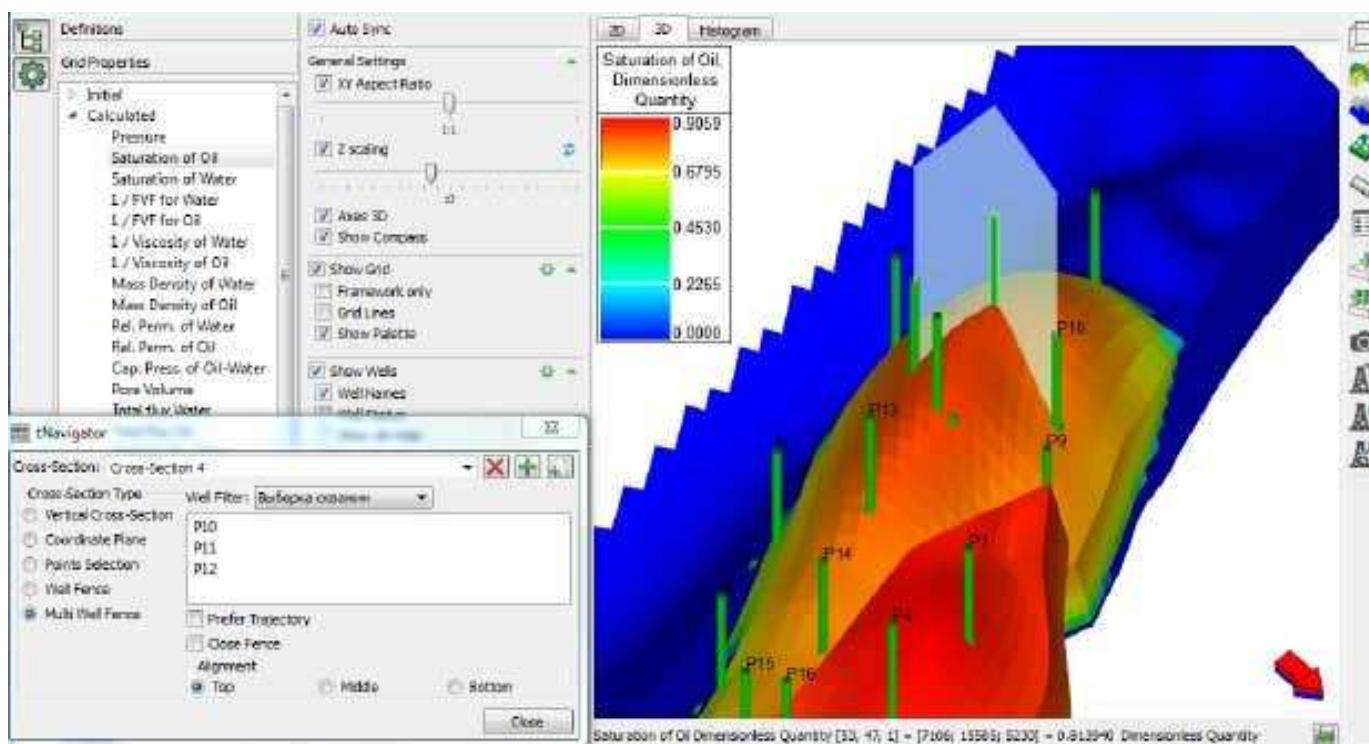


Figure 34. A Multi Well Fence.

8. Go to the 2D view. In the drop-down menu placed on the visualization settings tab select **Cross-Section–Cross-Section 5**.
9. **Fence Lines** is checked – figure 35. If this option is unchecked the vertical lines, corresponding to the fence points of cross-section, will be hidden.

Created cross-section can be visualized on current 3D-map. For more details see description of [3D map suboptions](#) and figure 56.

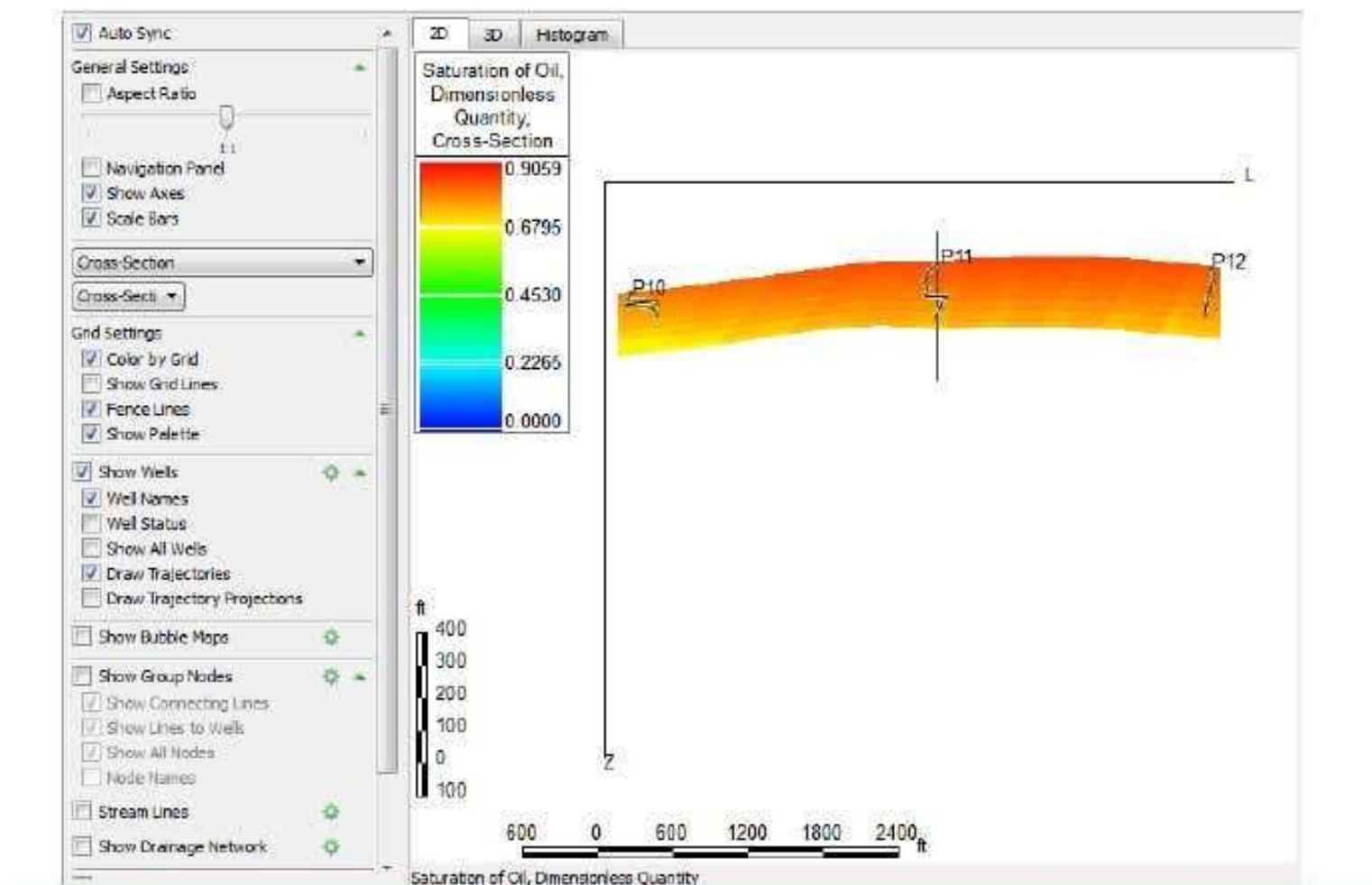


Figure 35. A multi-well fence in 2D.

5.7. Export

 Grid properties can be saved to a file and can later be re-loaded or added to the model via an include-file. The file will be saved to the model folder.

See the detailed description of all formats in the section [Export of grid properties](#).

5.8. Well, Groups and Network Filter. Stream Line Filter

A Use Well Filter.  The button with the well filter on.

The button **Well Filter** allows to:

1. **Create well filter** – section 5.8.1. If the filter is turned on, only the selected objects will be shown in all visualization; in the Graphs option (including Unified History Matching Results) only graphs and data for the selected objects will be accessible.
2. **Create filter for groups** – section 5.8.3. This filter allows to visualize only selected groups and connections between them in 2D.
3. **Create filter for surface network visualization** – section 5.8.4. This filter allows to visualize only selected nodes of surface network and branches between them in 2D.
4. **Create a Streamline Filter** – section 5.8.2. Using this filter allows to show all the wells of the model and visualize only streamlines for the wells selected by the streamline filter.

5.8.1. Well filter

The **Well Filter** dialog is shown in the figure 36. Using this dialog you can select wells one by one (left side of the dialog – **Single Well Selection**) or select multiple wells (right side of the dialog – **Multiple Well Selection**).

To select wells:

1. **Deselect All** there is no selected wells.
2. **Single Well Selection.** In the left side of the dialog, check the wells you need (or select all wells by clicking on **Select All**). You can find a well in the list of wells using the search line (start typing well's name or number in the search line).
3. **Multiple Well Selection.** Left-click on a group:
 - **FIPs** – selects wells in a certain fluid-in-place region.
 - **Current Historical and Calculated Well Types:** Producers, Injectors, Stopped, Shut, Not Present on This Time Step. **Accumulated** category means that the well have several statuses (e.g., a producer converted to an injector), so if you select **Accumulated**, the well will be selected as a producer and as an injector.
 - **Hist vs Calc. Low Rate Wells and Matched Wells** (these wells will be selected according to the settings used in the graphs Historical vs Calculated). Click **Select Only**. The wells selected will be added to the filter. You can **Add** wells to the filter or exclude wells from the filter by right-clicking on a group and press on **Select Only** or **Deselect**, respectively.

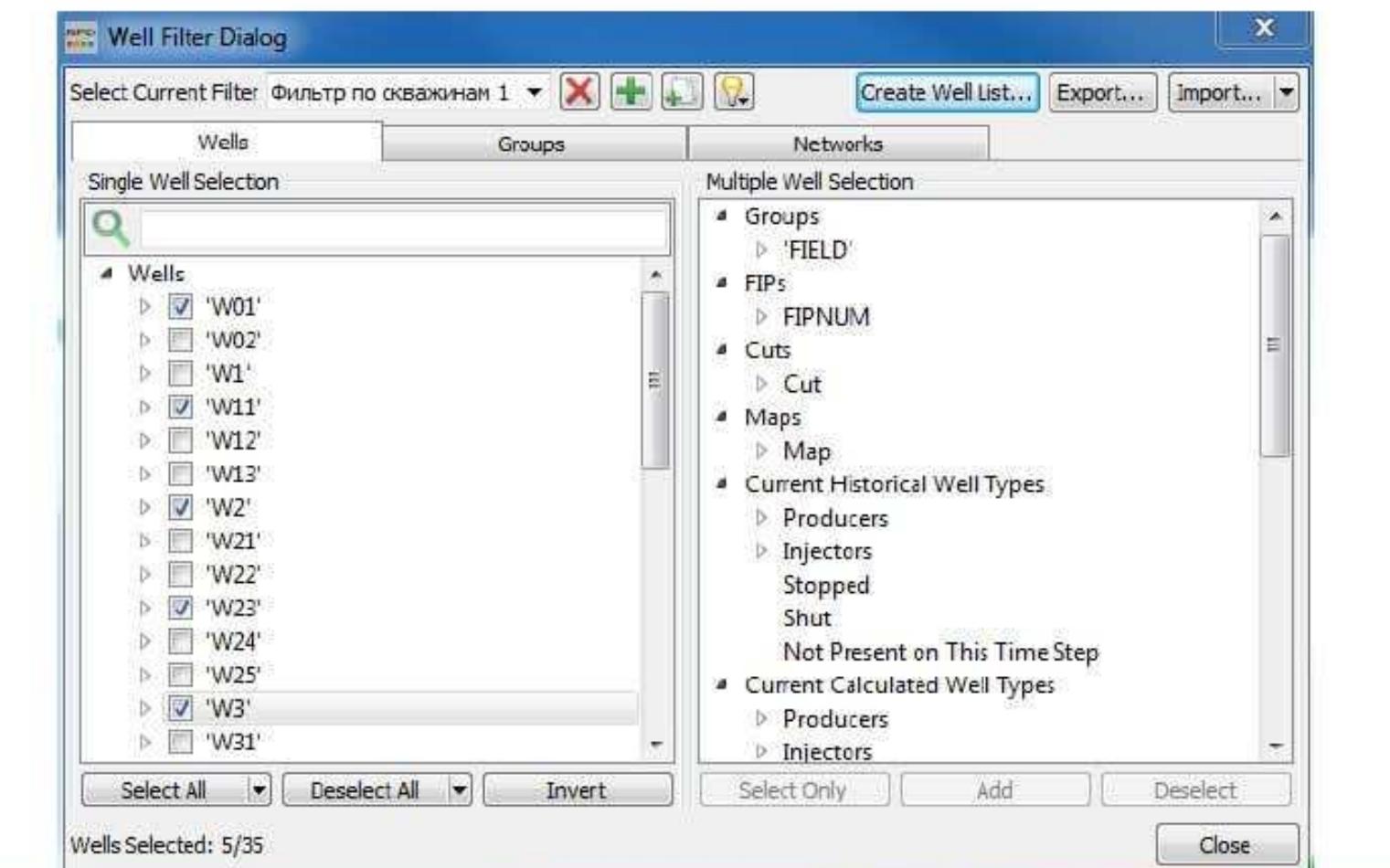


Figure 36. Creating a well filter.

4. After you include all the wells of interest in the filter, click **Close**.
5. Well filters will remain active when you close and re-open the model.

Buttons to work with filter.

- **Delete Filter.**
- **Create Filter.**
- **Duplicate Filter** the current well filter (create a new filter that only includes the wells selected in the current filter).
- **Use As Streamline filter** (the wells in the current filter will be included in the Stream Line Filter (a stream line filter description follows below)). **Use As Well Selection Filter.**
- **Create Well List.**

Currently selected wells will be saved as a well list (**WLIST**). Graphs for wells included this list are available as well. When list is created, the keyword **WLIST** (see 12.18.28) is writing to user-file in the **USER** folder.

Well list (**WLIST**) creating.

1. Check wells which will be included to a new list. Press **Create Well List**.
 2. Enter a list name. Press **OK**.
- **Export.** A list of wells in the filter will be saved to a text file. The file can be loaded to the model as a filter.
 - **Import.** A saved filter can be loaded from a file or from the clipboard.

5.8.2. Streamline Filter

This example is also presented in the training tutorial **2.1 How To Manage Waterflood**.

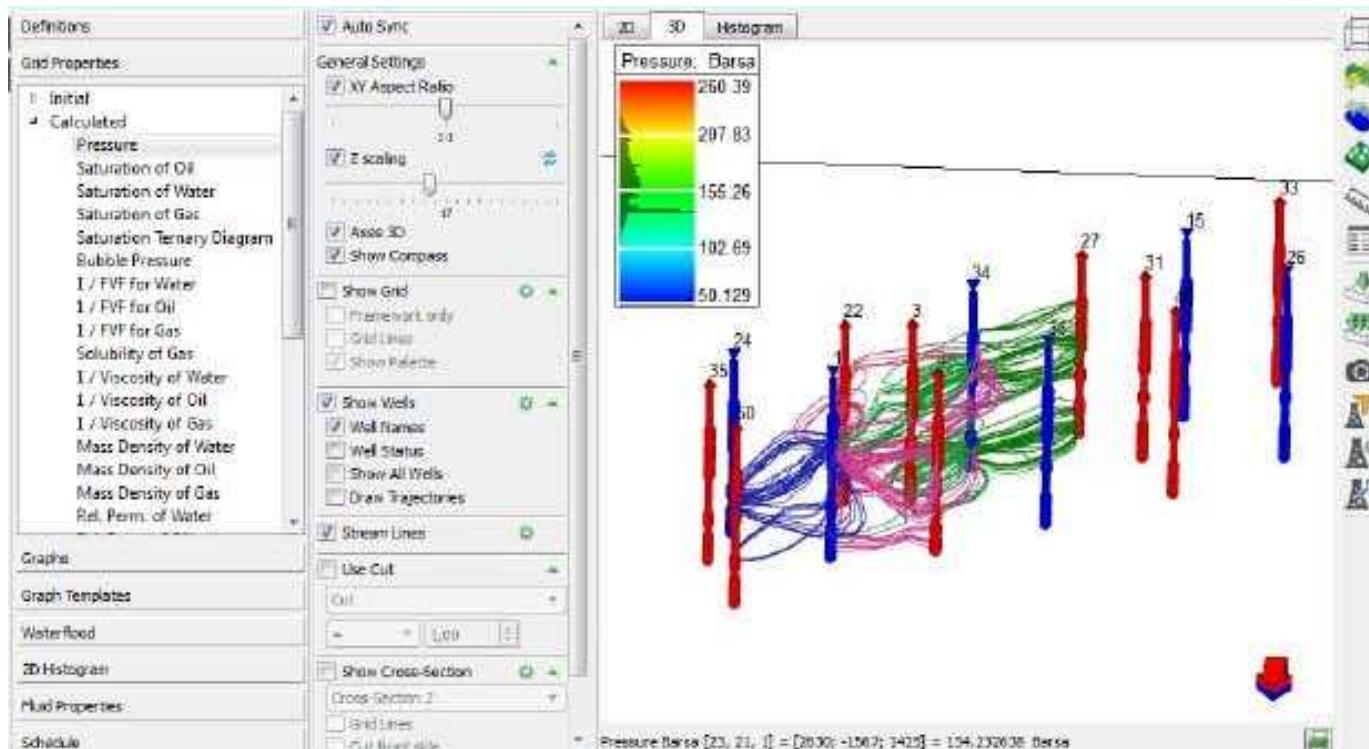


Figure 37. Streamlines for wells included in the Streamline Filter.

Creating a Stream Line Filter:

1. In the **Well Filter** dialog, select Current Filter – Streamline Filter. Select the required wells. For these wells streamlines will be visualized.
2. Select the Current Filter – **Well Filter 1 (or a different filter that includes wells required to be displayed)**. Well Filter 1 includes all wells. Click **Close** in the dialog.
3. Go to 3D view or 2D view of the model. Uncheck **Show Mesh** and check **Stream Lines**. Streamlines will be shown only for the selected wells—figure 37. However, all the wells of the model will be visible (according to Well Filter 1).
4. Go to 2D view. Right-click on the property near a well, and you will see a pop-up menu to define the **Stream Line Filter**. **Add This Well**. Selecting **Keep This Well Only** keeps only this well in the **Stream Line Filter**.

5.8.3. Group filter

The example of this functionality is shown in the training tutorial **1.11 How to use NETWORK**.

For models with group hierarchy defined via **GRUPTREE** (see [12.18.87](#)) its visualization is available in 2D view.

This filter allows to visualize only selected groups and connections between them in 2D. Switch to 2D view. Select **Groups** in the drop-down menu of **Nodes** located on the visualization settings tab to see the filter's action – picture [38](#).

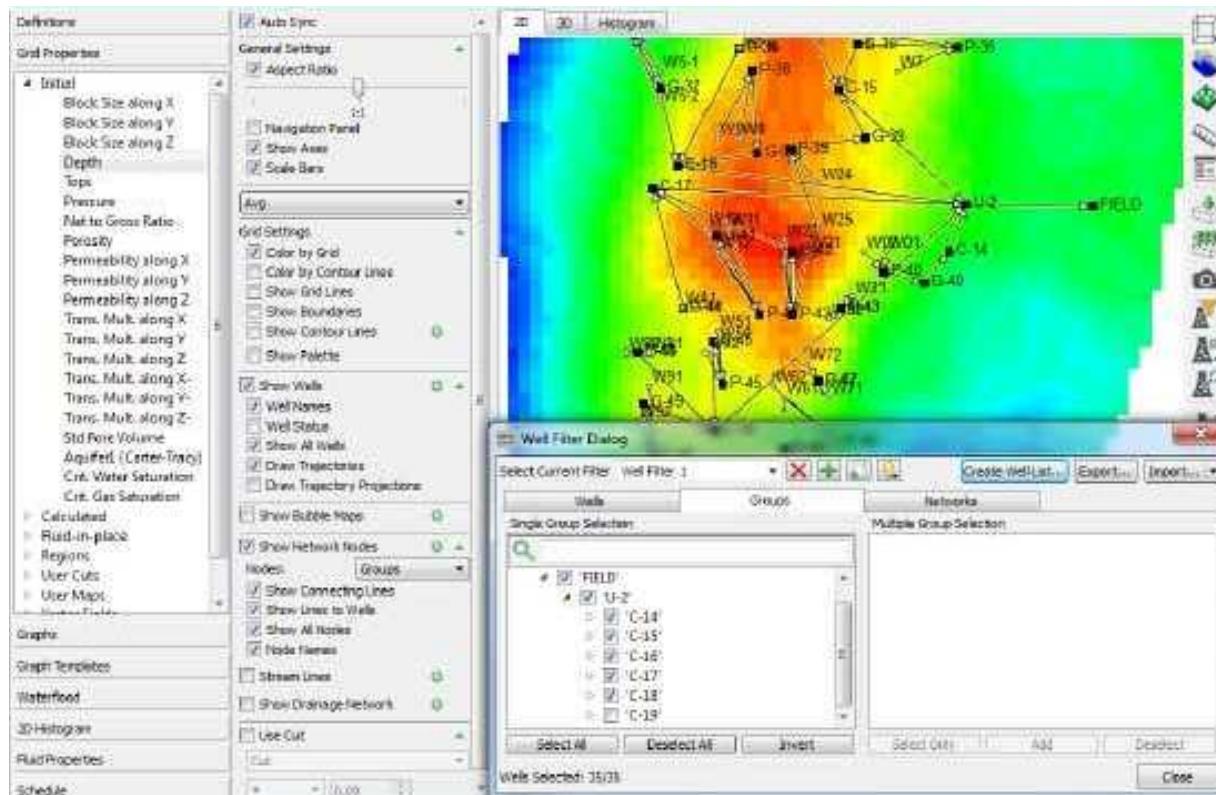


Figure 38. Group filter.

For groups with subordinate groups the following options are available via right mouse click on 2D visualizion:

- **Remove from filter.** The object will remove from filter and will not be visible.

5.8.4. Network filter

The example of this functionality is described in the training tutorial **1.11 How to use NETWORK**.

For models with surface network defined via **NETWORK** (see [12.1.87](#)) its visualization is available in 2D view.

The detailed description of these objects is in the section **NETWORK option. Automatic chokes. Compressors** of tNavigator user manual (document [tNavUserManualEnglish](#)).

This filter allows to visualize only selected network nodes and branches between them in 2D view. Select **Network Nodes** in the drop-down menu of **Nodes** located on the visualization settings tab to see the filter's action – picture 39.

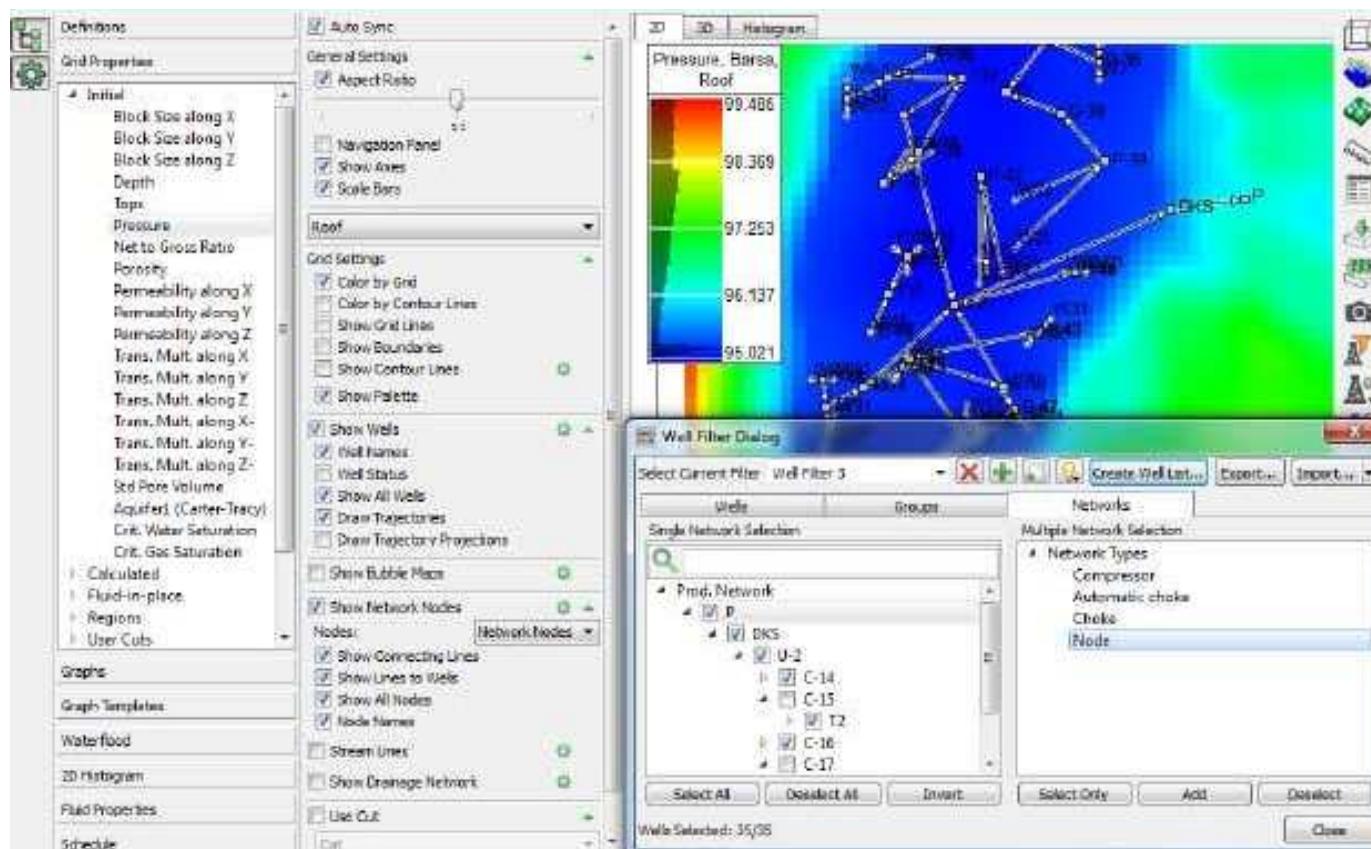


Figure 39. Network filter.

Multiple Selection is also available for the surface network visualization:

- **Compressor.**
- **Choke.**
- **Node.**

For objects with subordinate objects the following options are available via right mouse click on 2D visualization:

- **Remove from filter.** The object will not be visible on map.
- **Add all children to filter.** All child objects for selected one will be visualized in 2D view.
- **Remove from filter with children.** Selected object and all its "childs" will not be visible in 2D view.
- **Remove only children from filter.** All objects which are child to selected will not be visible in 2D view.

Also see the section [Surface network visualization](#). Bubble maps can be visualized for network nodes (e.g., node pressure, gas rate etc.).

5.9. Create Screenshot



Create Screenshot.

This option allows to save an image (e.g., properties, graphs) to raster, vector formats or print it. The Print dialog is shown in the figure 40.

Created image preview is on the right of tab **Print**. On the left the following parameters can be selected:

- **Page Layout:**
 - Best Fit;
 - Stretch Viewport.
- **Save to Raster File.** Raster file will be saved to the model's folder in .png or .jpg formats.
 - Width;
 - Height.
- **Save to Vector File.** Vector File will be saved to to the model's folder in .pdf format.
 - Page size. The following sizes are available: A3, A4, Letter;
 - Orientation: Landscape, Portrait.
- **Print to Microsoft XPS Document Writer.**
- **Print to.** To define print settings press the button **Configuration...**. The dialogue allowing to define page settings will appear. Number of copies is set in **Copies**.

On the tab **Caption** you can set a caption for your image.

- **Font;**
- **Position.** Above or below.

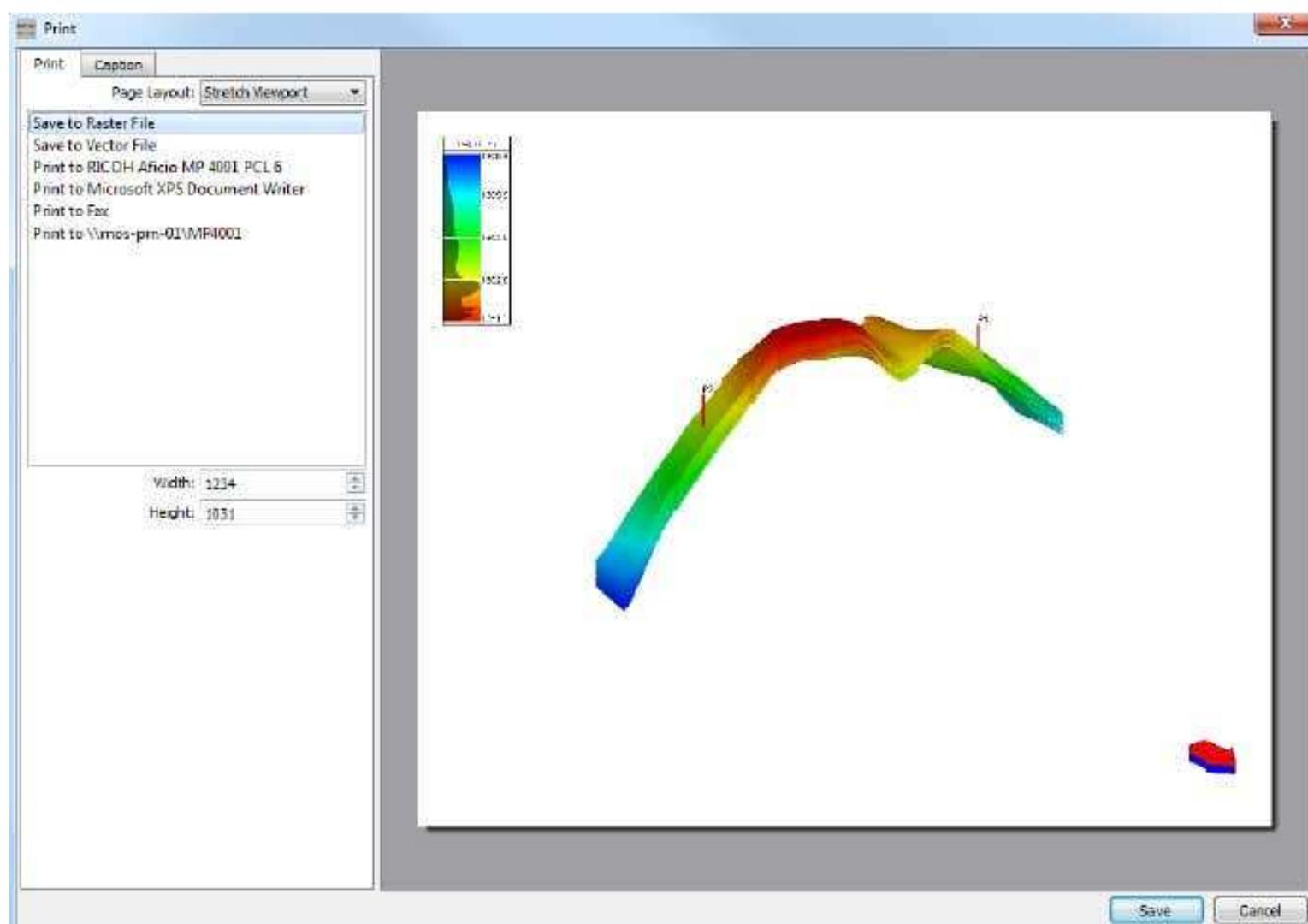


Figure 40. Screenshot dialog.

5.10. Well Actions

Well Actions.

This option allows to define settings for well creation and to export trajectories of wells. It is possible to define settings for **Single Well**, **Single Injector**, **Single Producer**, **Well Pattern**. Pressing **Alt+click** to add **Vertical Wells**, **Horizontal Wells**, **Well Pattern (Flooding Pattern)**.

See the training tutorial [How To Do Field Development Planning](#).

5.11. Find a Well or Connection

Find Well/Conn.

The dialog also has a feature allowing to find a well in a list. Just start typing the well's name or number. Wells with names coincide with the typed symbols will be highlighted with arrows in the visualization and moved to the top of the list – figure 41.

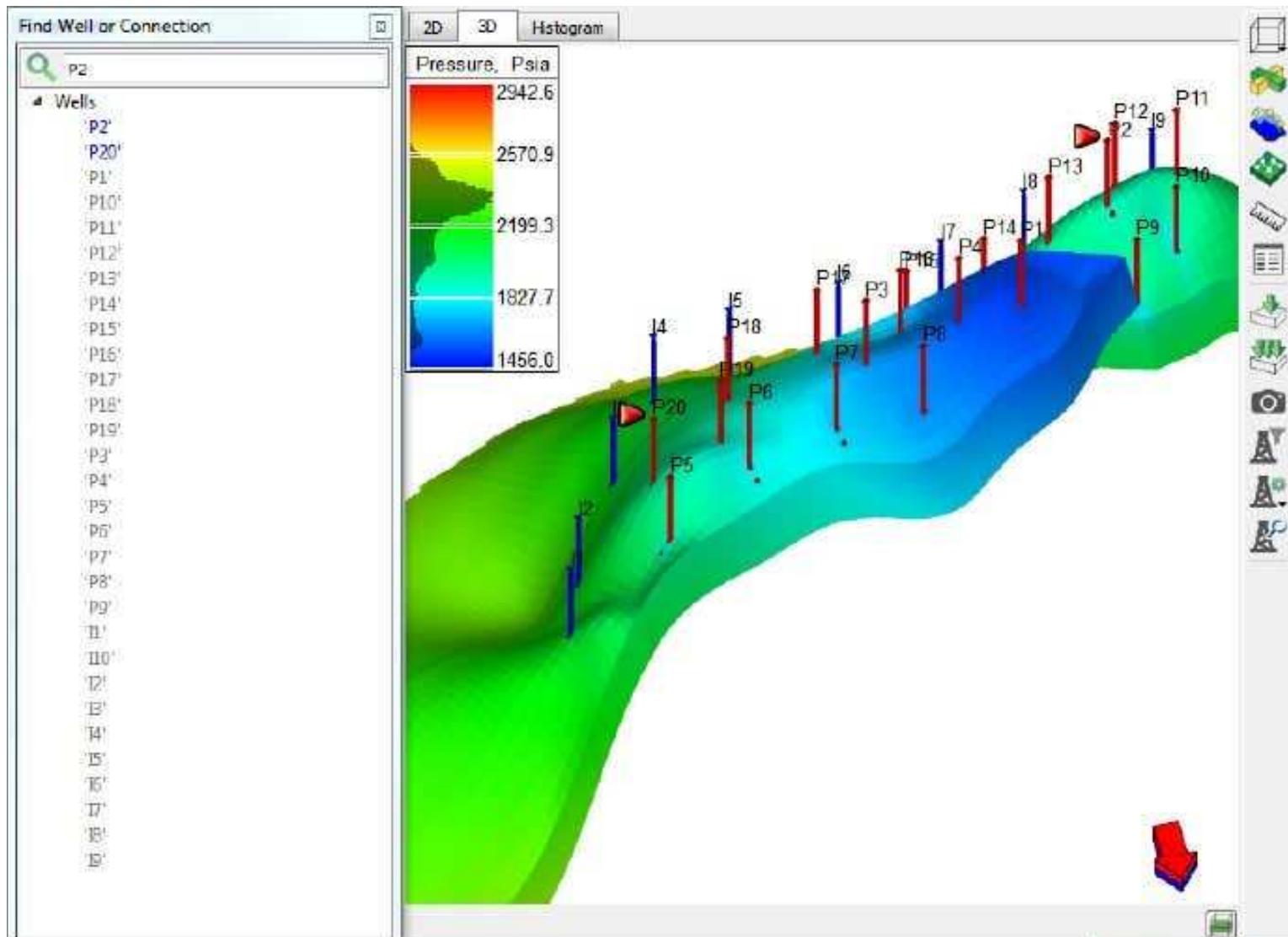


Figure 41. Find a well in the list or in the visualization.

5.12. Statistics

Statistics.

A statistics window will open for the current property, with the following data:

- **Entries** – the total number of active blocks;
- **Sum** – the sum of property's values for all the blocks;
- **Mean, RMS, Min, Max** – the property's mean, RMS, minimum and maximum values.

Using statistics it is possible to estimate total resources in the reservoir, average permeability and other parameters.

Data can be selected using a mouse and copied to a text editor (e.g., MS Excel).

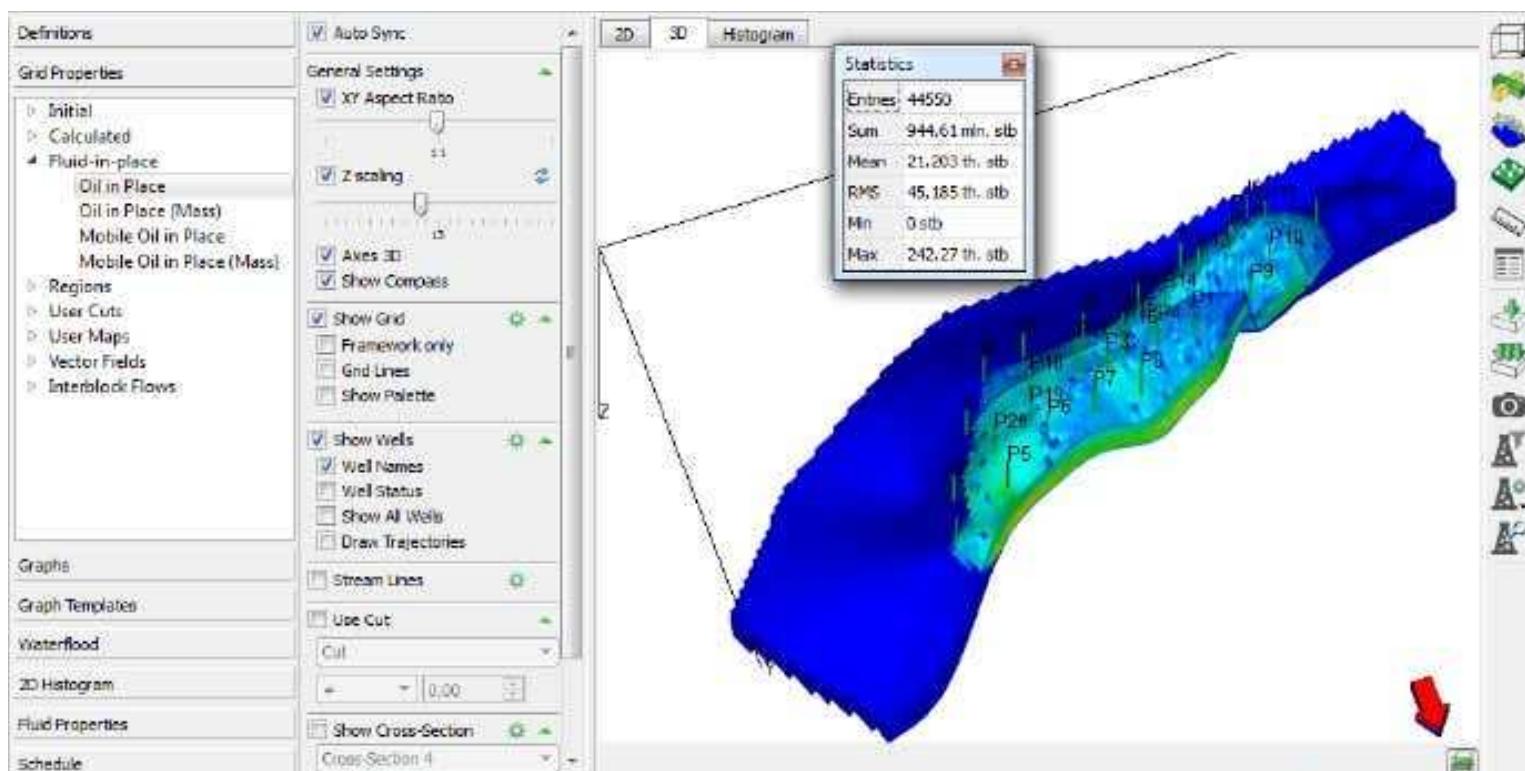


Figure 42. Statistics.

Statistics for several blocks only. This option can be used to estimated resources, e.g., in the selected area. This example is described in the training tutorial **1.1 How To Use tNavigator**.

If the User Cut is on, statistics will be shown for cut-selected blocks only:

1. Create User Cut filter.
2. Go to some property, for example, **Oil in Place (Mass)**.
3. Check **Use Cut**.
4. Only cut-selected blocks are shown.

5. Click **Statistics**.
6. **Statistics** is calculated only for the filter-selected blocks.

5.13. Well Selection

A Start Well Selection.

This button is available only in 2D view. This option will allow you to click and select wells that can be used as a Well Filter or for plotting graphs for the selected wells in the Graphs.

Create a Well Selection. There are 3 methods to create selection. The method is set in the dialog **Wells** which appears after A Start Well Selection button is pressed:

1. A **Select Wells One by One.** Select wells by clicking on them. Selected wells are marked by circles. To deselect well click on its icon again.
2. **Select Wells by Rectangle.** The rectangle area is set. Wells which are inside this area will be included in the selection. If it is necessary to select two or more well areas, specify them while holding **Shift** button. If it is necessary to remove some wells from a selection, get them in a rectangle while holding **Ctrl** button.
3. **Freeform Well Selection.** The freeform area is set. Well which are inside this area will be included in the selection (figure 43). If it is necessary to select two or more well areas, specify them while holding **Shift** button. If it is necessary to remove some wells from selection, get them in an area while holding **Ctrl** button.

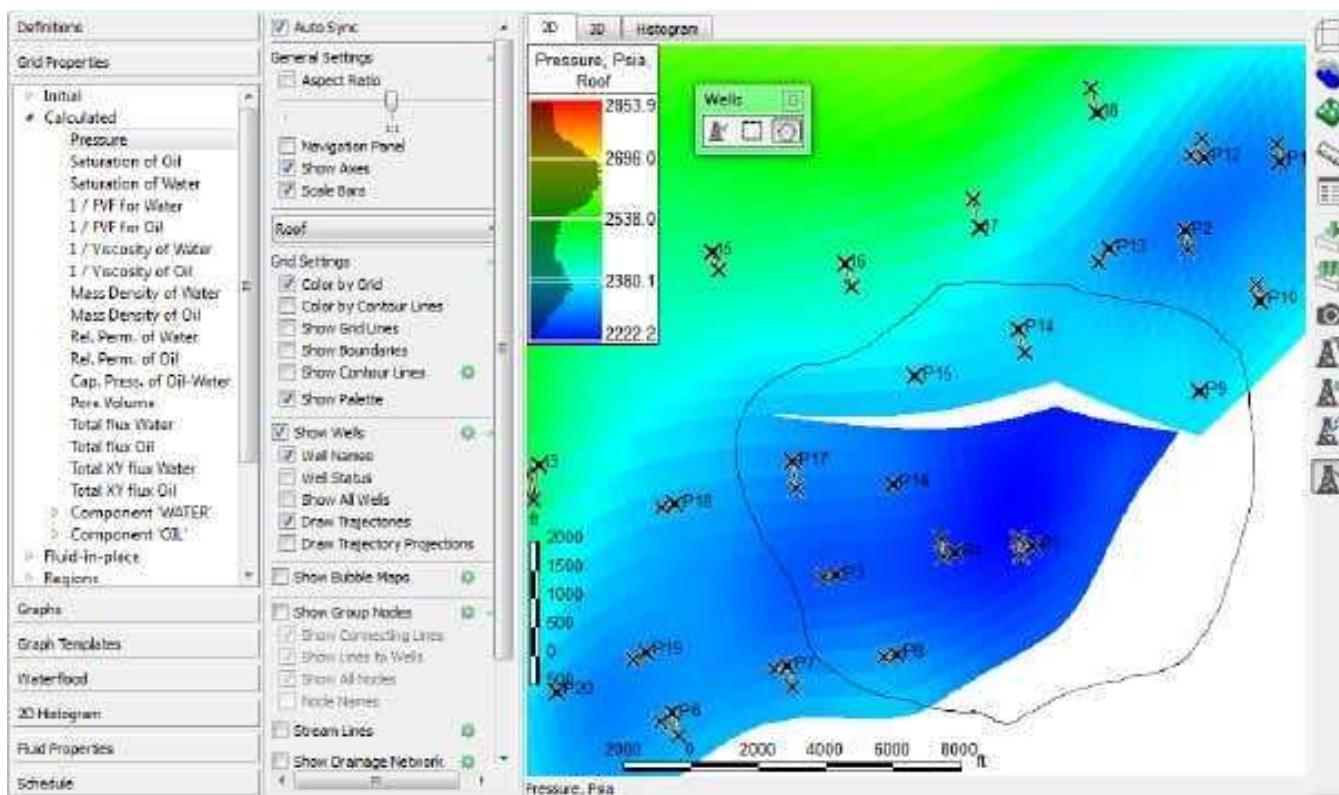


Figure 43. Freeform Well Selection.

By clicking button A Start Well Selection again you can (figure 44):

- Finish Wells selection;
- Clear Wells selection (this will clear all the wells from the selection);
- Export to Well Filter (the wells will be added to the Well Filter);
- Import from Well Filter (all the wells currently included in the well filter will be marked by circuses);
- Create well list. Currently selected wells will be saved as a well list (**WLIST**). Graphs for wells of this list are available too. When list is created, the keyword **WLIST** (see 12.18.28) is writing to user-file in **USER** folder;
- Keep Selected Only **Injectors, Producers, Stopped or Shut** wells;
- Remove from Selection **Injectors, Producers, Stopped or Shut** wells.

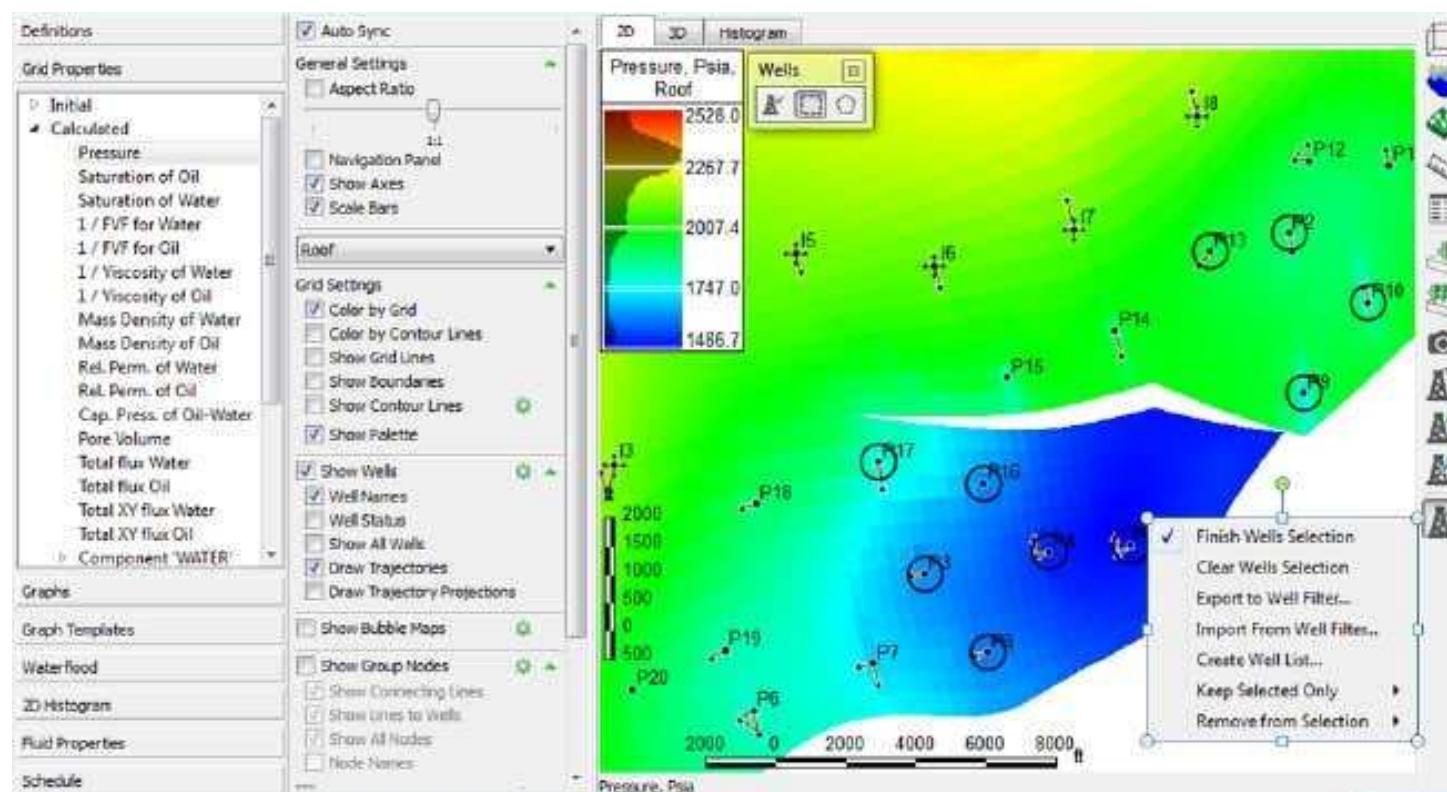


Figure 44. Well Selection.

Graphs for Selected Wells.

1. Click **Start Well Selection**.
2. Click **Create Graph Window** or press **Ctrl+N** to create a new **Quick Graph View**.
3. Select the wells required. They will be marked by circles.
4. In the Graph Window, a graph of the sum of the selected wells will be shown (Wells Group at the top of the window).

6. Grid Properties. General principles

View.

In the 3D view, the value of the selected parameter (formation top, porosity, pressure, oil saturation, reserves, etc.) is indicated in color.

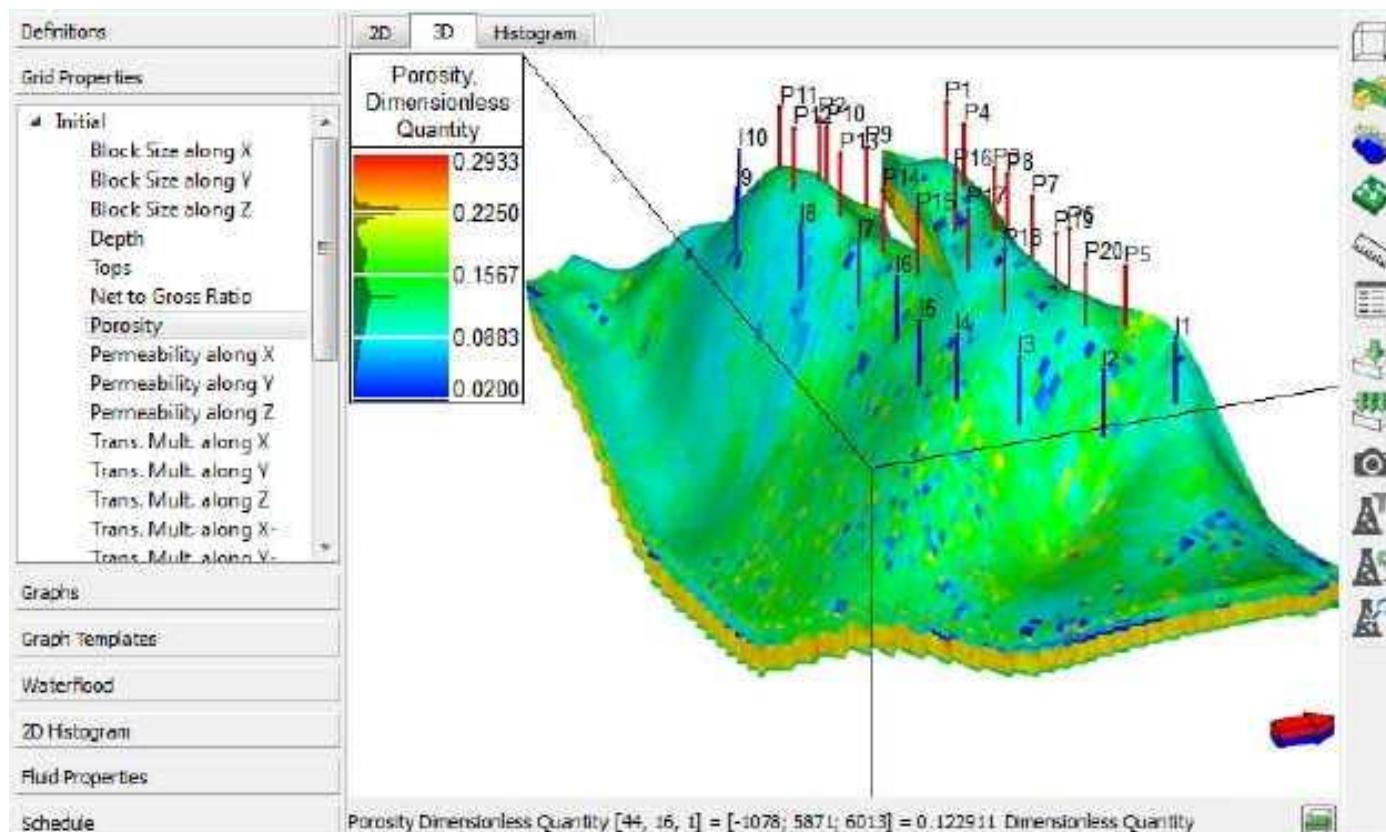


Figure 45. 2D. 3D. Histogram.

- Display options: 2D, 3D, Histogram – figure 45.
- You can rotate the image, by pressing and holding the left button of the mouse.
- You move the image within the window by pressing and holding the right button of the mouse.
- You can zoom the image using the mouse wheel or pulling the sliders **Zoom by Axes** on the visualization settings panel, **XY Aspect Ratio** and **Z Scaling**.
- Checking **Aspect Ratio** in 2D view will keep the actual aspect ratio and will not stretch the image through the entire visualization window.
- To the right of the visualization there are buttons to work with visualization.
- A vertical palette to the left of the visualization shows the color legend of the values of the shown parameter.

- Bringing the cursor to a block shows the block's information below the visualization: the block's coordinates in the grid (in blocks), the block's coordinates (in METRIC: m, FIELD: ft), and the value and the name of the shown parameter in the block (see figure 45).
- Checking **Auto Sync** synchronizes the zoom (or rotation) of image in two simultaneously opened windows of the model. To use this option, click  **Create New Window** on the horizontal tool panel. In the new window, you can view 2D or 3D visualization, zoom it synchronized with the view in the previously opened window.

Right click on Visualization.

Right-clicking on a block pops up the following menu:

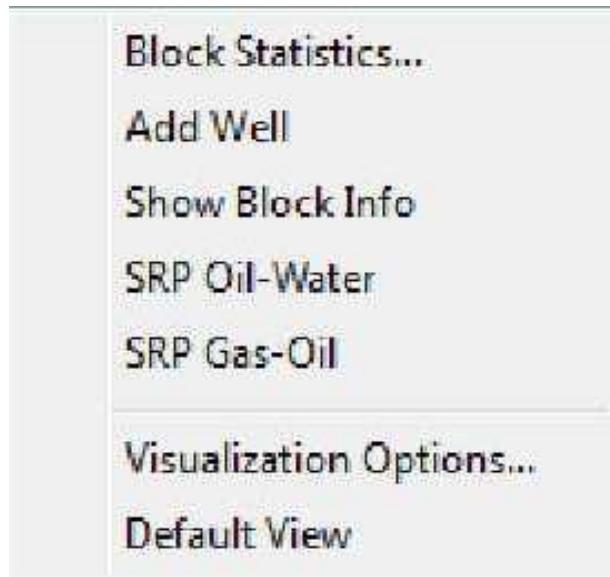


Figure 46. Right Click on Visualization Menu

- Selecting **Block Statistics** opens the **Statistics for Block [Block Number]** window. The values of all parameters (initial and calculated) for this block are shown. [The block number] in X, Y, Z axes is shown on the left. To go to statistics for another block, you can select other block's numbers in X, Y, Z on the left. To save **Block Statistics** select the required column(s) or line(s) by clicking on them and press **Ctrl+N** (Copy). To select the whole table click on its top left corner. The data can be pasted into an Excel worksheet using the combination **Ctrl+V** (Paste).
- Selecting **Add Well** opens the Add Well Dialog.
- Selecting **Show Block Info** is equivalent to double-clicking on the block without the well. This moves you to the **Graphs** option, Block Info sub-option. The graphs of the selected block's parameters will be shown.
- Selecting SRP Oil-Water (SRP Gas-Oil) moves you to **Fluid Properties, SRP Oil-Water (SRP Gas-Oil)** sub-option for this block (scaled relative permeabilities).

- Selecting **Visualization Options** moves you to the Visualization Options dialog.
- Selecting **Default View** returns the property to the default view (any user-defined zoom and movement of the visualization will be canceled).

6.1. Palette

The palette sets the correspondence between the values of the current parameter and colors. The palette is a vertical strip to the left of the visualization of parameter. You can move the palette by its top (with the property's title). Colors on the palette are separated by horizontal lines (each line is the start of a color). Between the lines, colors are linearly interpolated. The values corresponding to the horizontal lines are shown near each line of the palette. Color's lines can be moved pressing on the left button of the mouse. Right-clicking on the palette will display the following menu – figure 47:

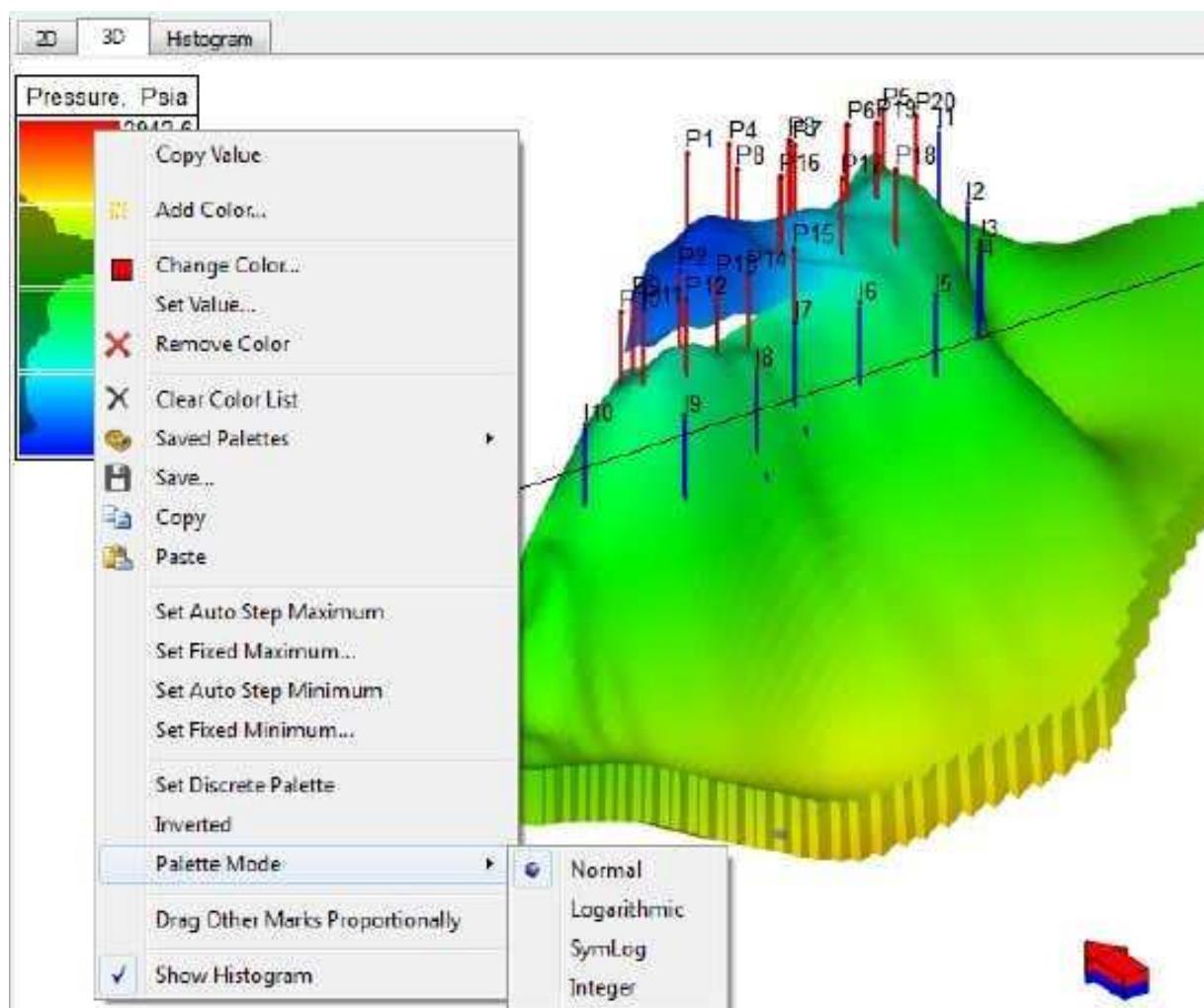


Figure 47. Palette.

- **Copy Value.** The value clicked is a number. You can paste that number into a text file by pressing **Ctrl+V**.
- **Add Color.** Add a color to the current item.
- **Change Color.** Change the nearest color.
- **Set Value.** Set the user's value to the selected color.
- **Remove Color.** Remove the nearest color.

- **Clear Color List** (clears all the colors and values, except the maximum and the minimum; all colors become gray).
- **Saved Palettes.** Open a 5-colors default palette, a 7-colors default palette, k-colors palette (palette with arbitrary colors number; k is set by user) or palettes saved by user. To rename or delete a palette press the right mouse button and select corresponding option (picture 48).

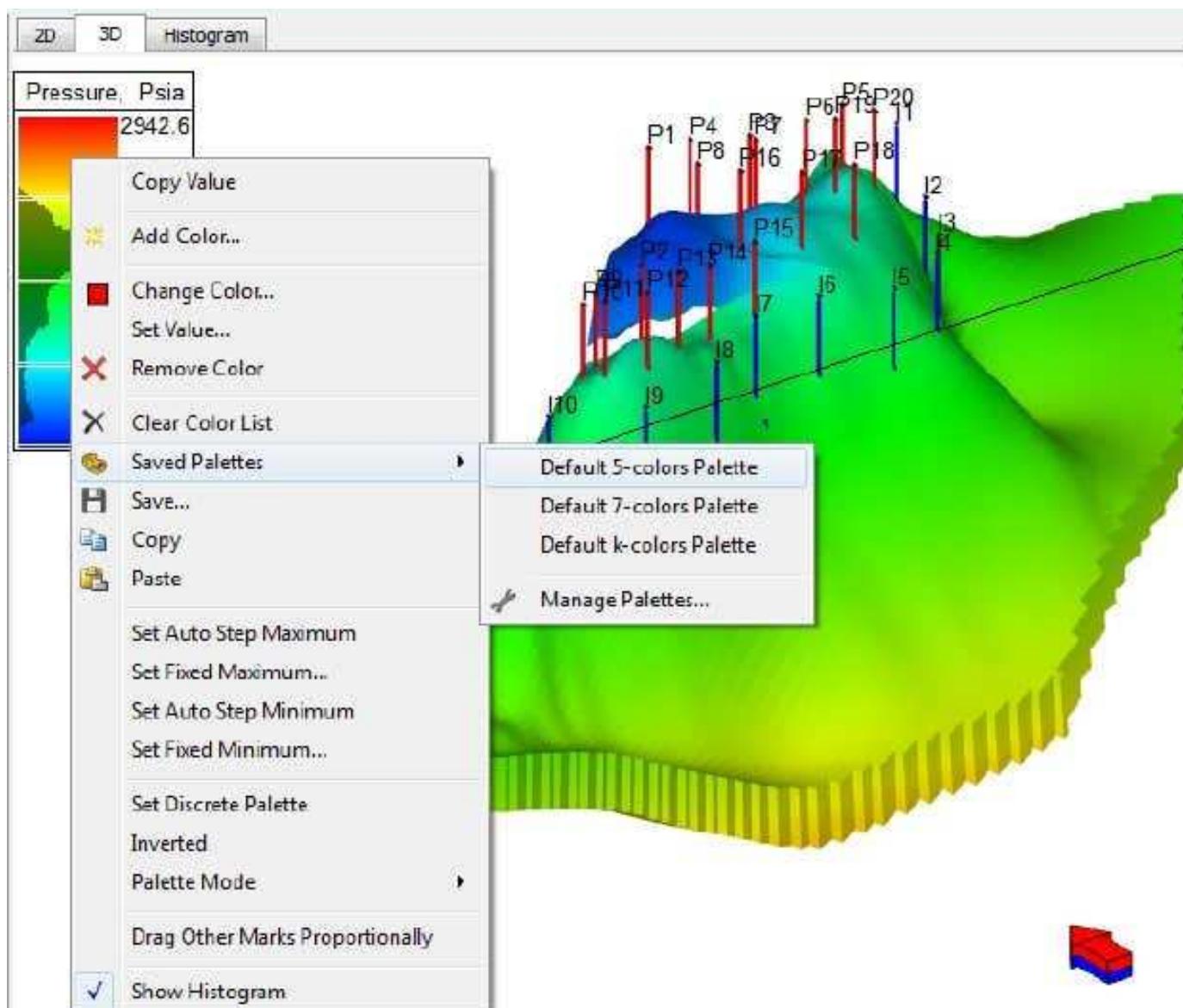


Figure 48. Palette management.

- **Save.** Saves the current palette.
- **Copy.** The current palette is copied and can be used in the different tNavigator window (click **Paste Palette**).
- **Paste.** The palette will be pasted.

The palette includes 4 options for setting the maximum and/or the minimum: **Set Auto Step Maximum**, **Set Auto Maximum for All Calculated Steps**, **Set Fixed Maximum** and **Set Fixed Minimum**:

- **Set Auto Step Maximum** (the default setting for the property is to calculate the maximum of a parameter for all steps – cumulative maximum, i.e. the maximum increases constantly, and the palette's borders expand. If this option is selected the cumulative maximum, re-calculated at each time step, is shown in 2D or 3D view. To go back to the cumulative maximum computation, right-click the palette and select **Auto Maximum for All Time Steps**);
- **Set Fixed Maximum** (this opens the window for setting a new maximum; after setting a fixed maximum you can change it by selecting **Change Current Maximum**; to come back to auto maximum computation, right-click on the palette and select **Set Auto Step Maximum**).
- **Set Auto Step Minimum** (the default setting for the view is to calculate the minimum of a parameter for all steps – cumulative minimum, i.e. the minimum decrease constantly, and the palette's borders vary. If this option is selected the minimum, re-calculated at each time step, is shown in 2D or 3D view. To go back to the cumulative minimum computation, right-click on the palette and select **Set Auto Minimum for All Time Steps**).
- **Set Fixed Minimum** (this opens the window for setting a new minimum; after setting a fixed minimum you can change it by selecting **Change Current Minimum**; to come back to auto maximum computation, right-click on the palette and select **Set Auto Step Minimum**).
- **Set Discrete Palette** (this will cancel linear interpolation between colors and will show a contour line analogue; to come back to a continuous palette, right-click palette and select **Set Continuous Palette**). In the discrete palette (see figure 49) property's colors corresponding to a property's values vary from the minimum (indicated by the horizontal line at the bottom of the palette's color) to the maximum (indicated by the horizontal line at the top of the palette's color). In a continuous palette every value of the parameter corresponds to a color shade of the palette.

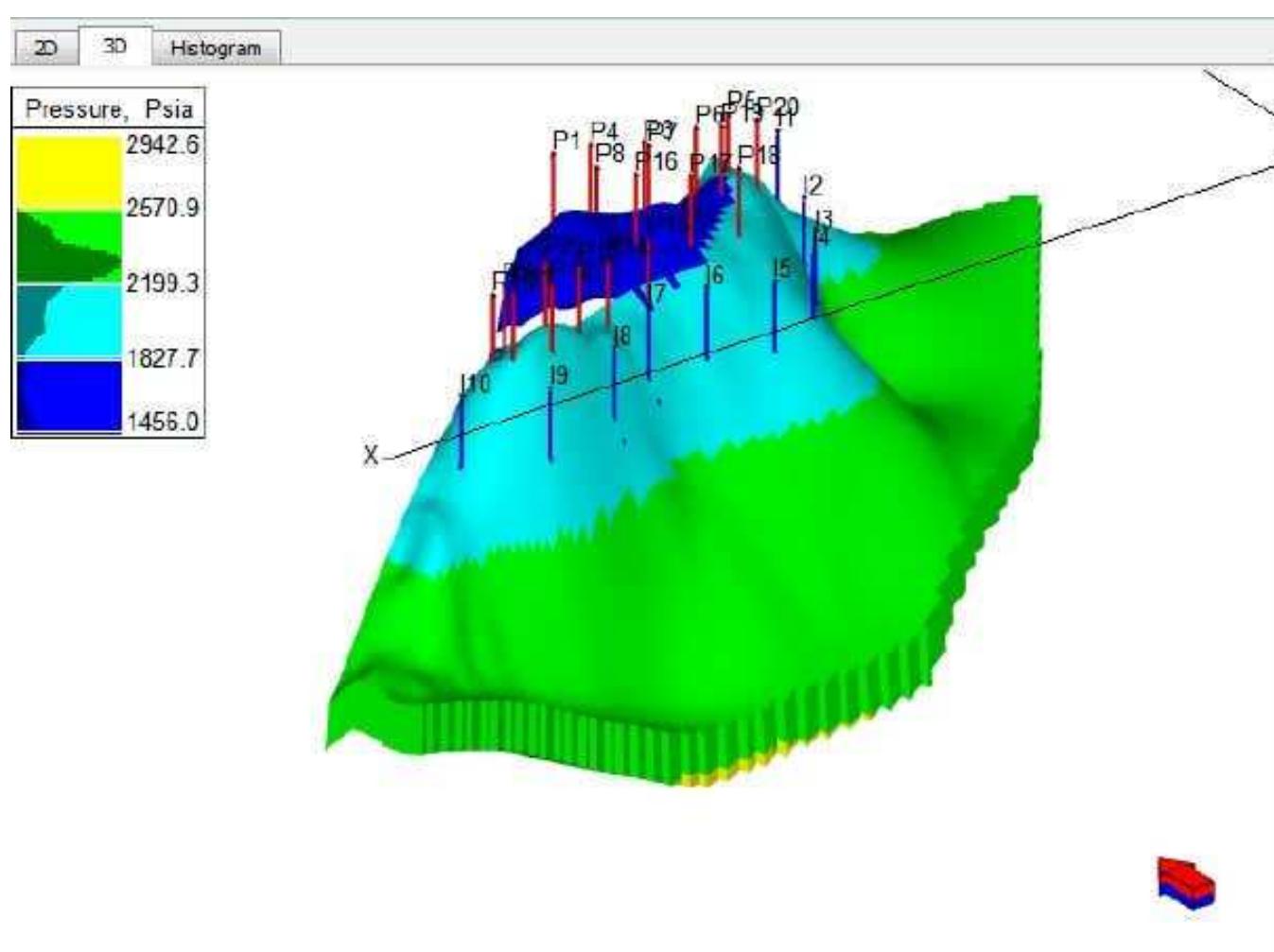


Figure 49. Discrete Palette.

- **Inverted Palette.** Red color marks maximum parameter's values, and blue color marks minimum property values. In an inverted palette (select **Inverted Palette**), maximum values will be inverted to blue color, and the minimum values will be inverted to red color. Inverted Palette is the default setting for the initial grid properties **Tops (Grid Properties. Initial)**;

- **Palette Mode**

- **Normal.** Default palette.
 - **Logarithmic Palette** (colors corresponding to the positive values are distributed logarithmically, while the negative values correspond to 0). This is the default setting for the PermX, PermY, PermZ distributions—**Grid Properties.Initial**).
 - **SymLog.** Logarithmic change of palette's colors is carried out for positive and negative values separately.
 - **Integer.** Palette values are integers. Property's values in blocks are rounding using math rules.
- **Drag other marks proportionally.** When you move any palette's mark others will move automatically. The direction of other marks coincide with the direction of the selected mark. Shifts of other marks are proportional to the shift of the selected one: the smallest shift corresponds to the mark located at the highest distance from the selected one.
- **Show Histogram.** The distribution of property's values will be shown on the palette.

6.2. Local Grid Refinements (LGR)

If there are local grid refinements in the model, the LGRs will be shown in 2D, 3D (keywords: **LGR** (see 12.1.84), **CARFIN** (see 12.2.89), **REFINE** (see 12.2.90), **ENDFIN** (see 12.2.91), **WEL-SPECL** (see 12.18.4), **COMPDATL** (see 12.18.7), **NXFIN** (see 12.2.92), **NYFIN** (see 12.2.92), **NZFIN** (see 12.2.92), **HXFIN** (see 12.2.93), **HYFIN** (see 12.2.93), **HZFIN**, see 12.2.93).

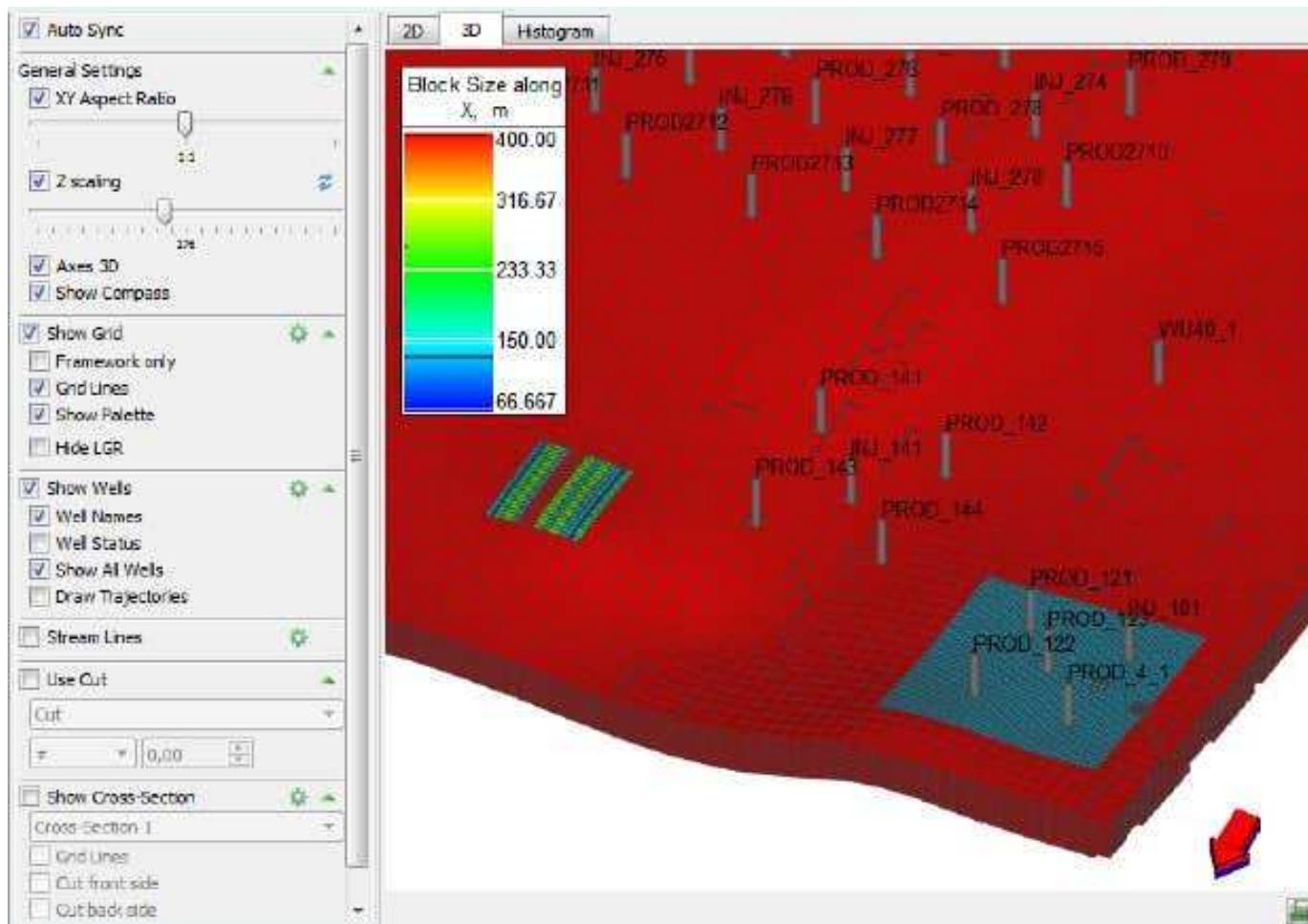


Figure 50. Two LGR areas in 3D.

On the visualization settings panel, there is **Hide LGR** option. Checking it will hide the LGR.

You can add wells to LGR areas in the same way you add wells to the areas without grid refinement.

See the training tutorial **7.2 Local Grid Refinement LGR**.

6.3. Properties for dual porosity model

For models with dual porosity and dual permeability (keywords DUALPORO (see 12.1.79), DUALPERM, see 12.1.80) all properties are visualized on two tabs **Matrix** and **Fracture** – see the picture 51. Switch between these tabs to see the corresponding properties.

The detailed description of dual porosity and dual permeability models is given in the section of tNavManualEnglish Dual porosity.

See also the training tutorial **7.3 Dual porosity**.

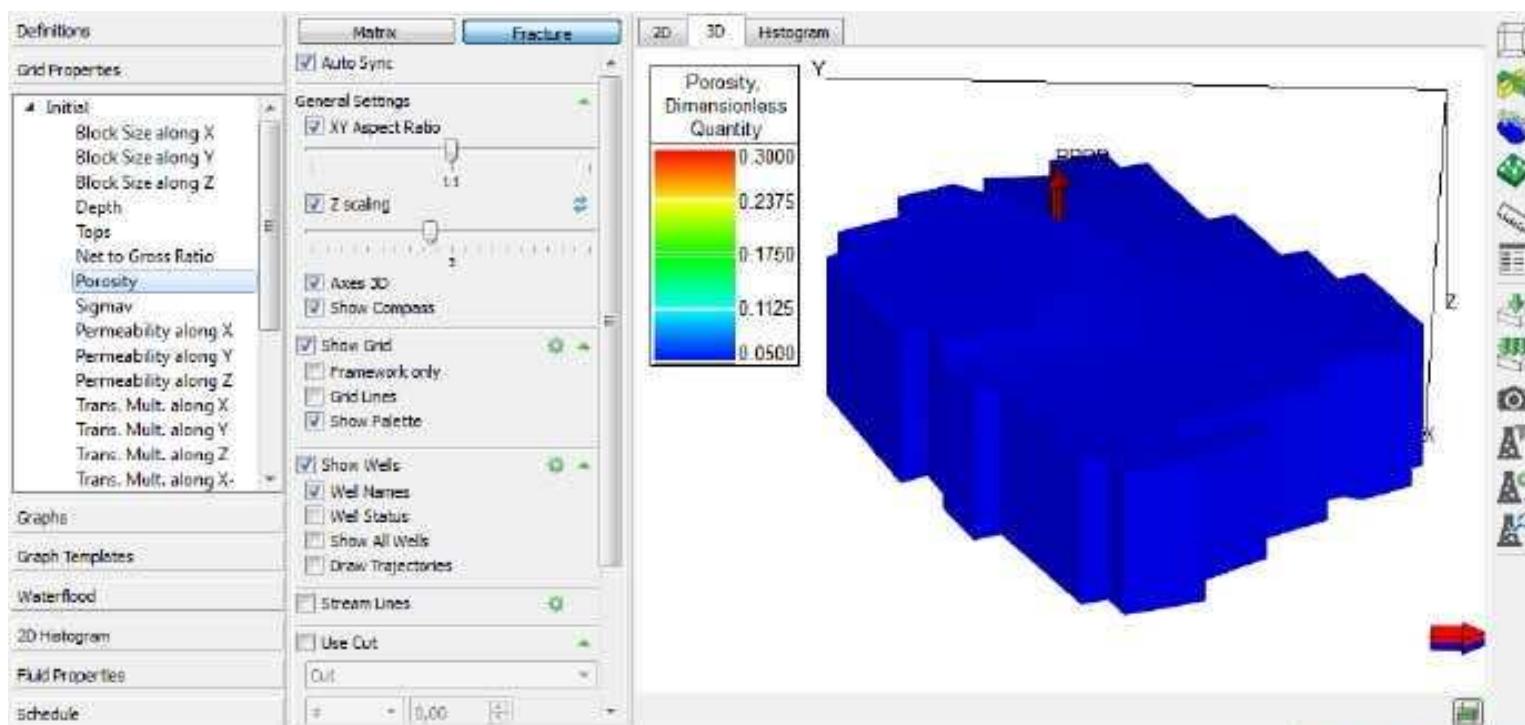


Figure 51. Dual porosity model.

Note. Properties visualization with the keyword **DPNUM** (see 12.2.67).

If the keyword DPNUM (see 12.2.67) is defined in the model with a dual porosity and a single permeability (define the regions which should be considered as regions with single porosity). These blocks are visualized on the tab **Fracture** (and are not visualized on the tab **Matrix**). Such visualization is the result of these blocks' behaviour is corresponds to the fracture's blocks: the flow is possible between these blocks located in the single porosity regions. Moreover, these blocks can be perforated.

- Total voidage replacement coefficient (re-injection ratio under reservoir conditions) is calculated as Reservoir Volume Injection Total (rm^3) divided by Reservoir Volume Production Total (rm^3);
- Ratios (H) – historical values of ratios listed above;
- oil recovery factor (is calculated as initial oil-in-place minus current oil-in-place divided by initial oil-in-place) (%);
- number of producers currently flowing;
- number of injectors currently flowing;
- number of new wells;
- material balance error (METRIC: m^3 , FIELD: stb);
- oil material balance error (METRIC: m^3 , FIELD: stb);
- water material balance error (METRIC: m^3 , FIELD: stb);
- gas material balance error (METRIC: m^3 , FIELD: Mscf);
- Mass of water (METRIC: kg, FIELD: lb);
- Mass of oil (METRIC: kg, FIELD: lb);
- Mass of gas (METRIC: kg, FIELD: lb);
- Number of moles for water;
- Number of moles for oil;
- Number of moles for gas;
- water volume (METRIC: m^3 under reservoir conditions, FIELD: b under reservoir conditions);
- oil volume (METRIC: m^3 under reservoir conditions, FIELD: b under reservoir conditions);
- gas volume (METRIC: m^3 under reservoir conditions, FIELD: b under reservoir conditions);
- compensation (a graph of production vs injection, in percentage points).

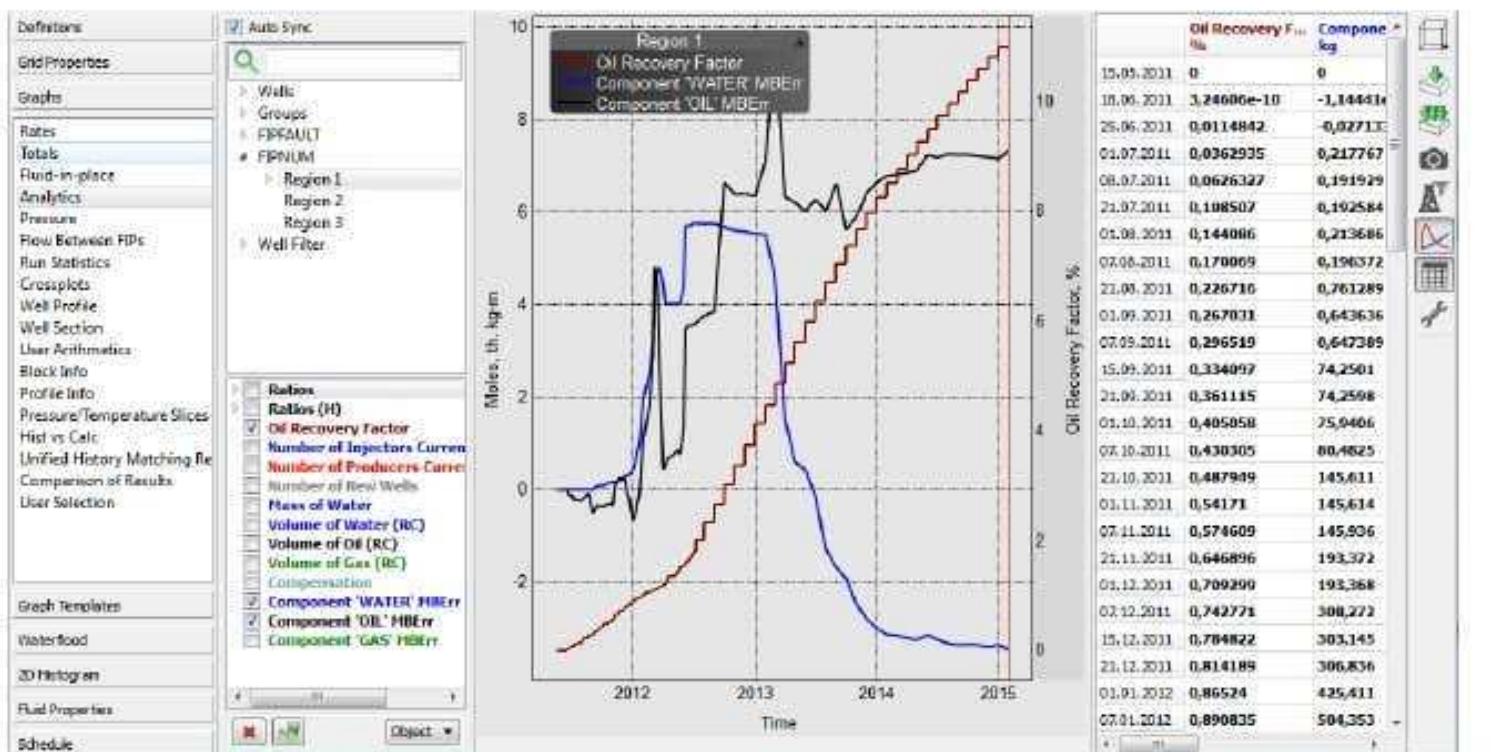


Figure 114. FIPNUM 1 Analytical Graphs.

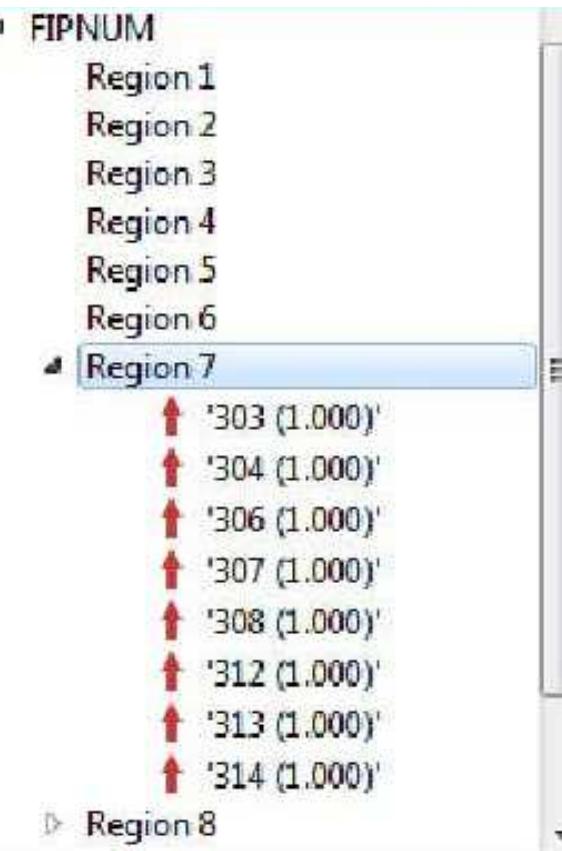


Figure 115. Well Tree.

In the figure 114 oil recovery factor and injection ratio graphs are checked. The graph uses two scales. The left scale (highlighted in gray) is for the recovery factor, the right one, for the material balance errors.

Every FIP region has a well tree comprising all the wells in the region (see figure 115). In the Object Items selection dialog, the well name is followed by a number in parentheses.

sis – the number of FIP regions in which this well is found. For example, Well A1 (1.000) means that Well A1 is found in one FIP region; Well A2 (2.000) means that Well A2 is found in two FIP regions.

Segments object.

A segment well structure allows to describe a flow more accurately. A well is split into parts – segments, each segment has its own set of parameters. Links to the keywords which specify multisegment wells are in the section Multisegment well of tNav User Manual.

The following graphs are available:

- Ratios:
 - gas-oil ratio (METRIC: m^3/m^3 , FIELD: Mscf/stb);
 - oil-gas ration (METRIC: m^3/m^3 , FIELD: stb/Mscf);
 - gas-water ratio (METRIC: m^3/m^3 , FIELD: Mscf/stb);
 - water-gas ratio (METRIC: m^3/m^3 , FIELD: stb/Mscf);
 - water-oil ratio;
 - oil-water ratio;
 - liquid-gas ratio (METRIC: m^3/m^3 , FIELD: stb/Mscf);
 - gas-liquid ratio (METRIC: m^3/m^3 , FIELD: stb/Mscf);
 - watercut;
- Water flow velocity (METRIC: m/s , FIELD: ft/s);
- Oil flow velocity (METRIC: m/s , FIELD: ft/s);
- Gas flow velocity (METRIC: m/s , FIELD: ft/s);
- Water holdup fraction;
- Oil holdup fraction;
- Gas holdup fraction;
- Segment THP Length (METRIC: i ; FIELD: ft);
- Segment BHP Depth (METRIC: i ; FIELD: ft);
- Segment Diameter (METRIC: i ; FIELD: ft);
- Segment Roughness (METRIC: i ; FIELD: ft);
- Segment Area (METRIC: i^2 ; FIELD: ft^2);
- Segment Volume (METRIC: i^3 ; FIELD: ft^3).

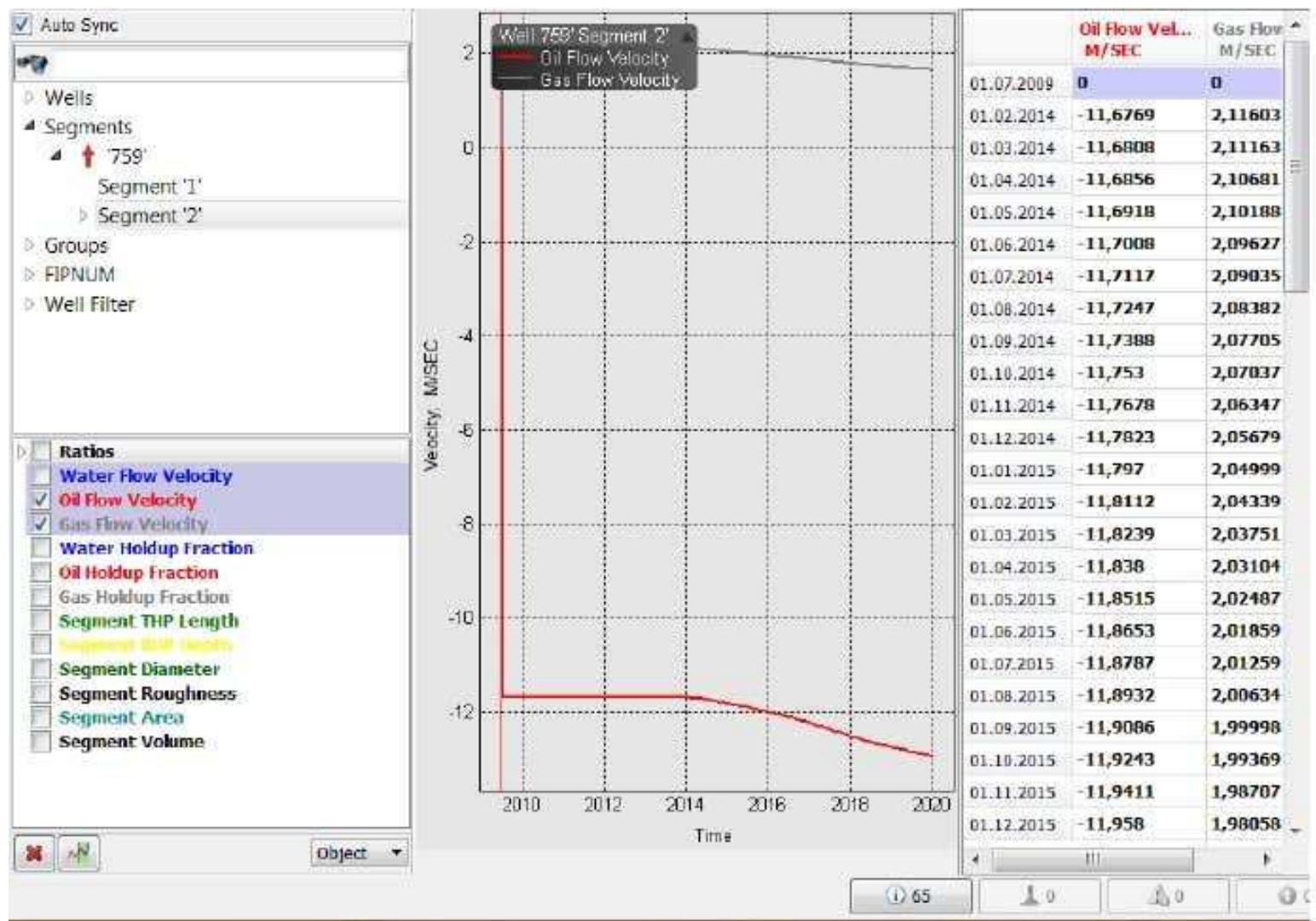


Figure 116. The Oil flow velocity in the segment.

The object **Neworks** (nodes of surface network).

See the detailed description in the section NETWORK option. Automatic chokes. Compressors of UserManual.

The following graphs are available:

- Ratios (gas-oil, watercut and other);
- ALQ (Artificial Lift Quantity);
- VFP table number.

8.5. Pressure

For the object items **Group** and **Field**, this tab displays the graphs specified below:

- Average pressure (reservoir pressure) weighted by pore volume (METRIC: bar, FIELD: psi);
- Network node pressure (see the keyword **NETWORK**, see 12.1.87) (METRIC: bar, FIELD: psi);
- Loaded pressure (METRIC: bar, FIELD: psi). It can be loaded using the button  on the right panel, see the section 7.1

The pressure weighted by pore volume is calculated as follows: pressure in each block is multiplied by the pore volume of the block. Then the products of all the blocks are summarized, and the total is divided by the sum of all the blocks' pore volumes (wpv - Weighted by Pore Volume).

If Eclipse or Tempest MORE results have been loaded (if available, Eclipse or Tempest MORE results get loaded by default - tNavigator's General Settings), Eclipse or Tempest MORE results will also be accessible, with names designated as described above ([E] for Eclipse and [M] for Tempest MORE). Example: **Weighted Average Pressure [E]** means the weighted average pressure calculated by Eclipse.

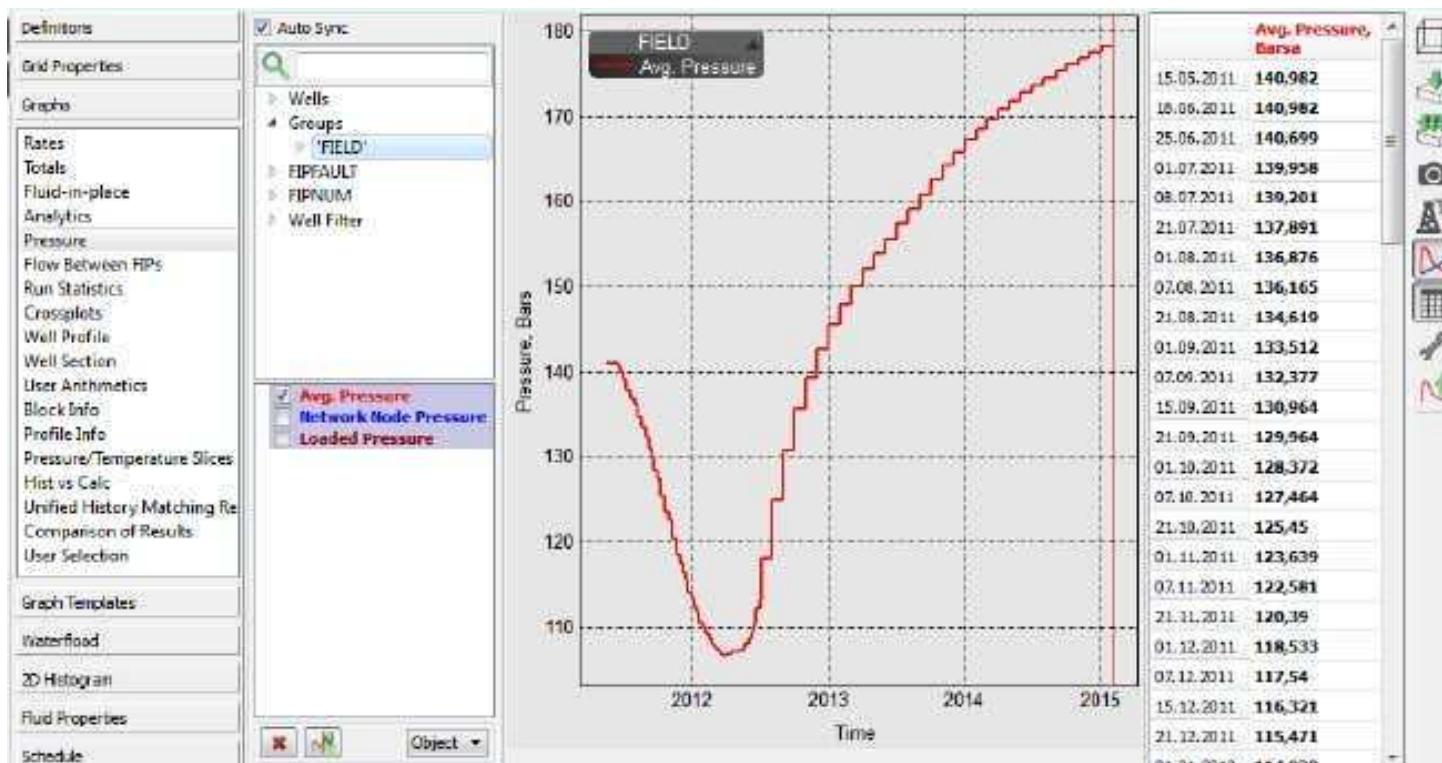


Figure 117. Reservoir Average Pressure Graph.

For the object item **Well**, you can view the graphs listed below:

- Bottom hole pressure (METRIC: bar, FIELD: psi);
- Bottom hole pressure (H) – historical bottomhole pressure (METRIC: bar, FIELD: psi);

- Bottom hole pressure target and history – assigned historical bottomhole pressure (pressures set by the keyword **WCONHIST** (see 12.18.37) for producers and by the keyword **WCONINJH** (see 12.18.41) for injectors) (METRIC: bar, FIELD: psi);
- Bottom hole pressure target – assigned bottomhole pressure (pressures set by the keyword **WCONPROD** (see 12.18.36) for producers and by the keyword **WCONINJE** (see 12.18.38) for injectors, **WELTARG**, see 12.18.53) (METRIC: bar, FIELD: psi);
- Tubing head pressure (METRIC: bar, FIELD: psi);
- Tubing head pressure target and history – assigned tubing-head pressure (pressures set by the keyword **WCONHIST** (see 12.18.37) for producers and by the keyword **WCONINJH** (see 12.18.41) for injectors) (METRIC: bar, FIELD: psi);
- Tubing head pressure (H) – historical tubing-head pressure (METRIC: bar, FIELD: psi);
- Tubing hole pressure target (METRIC: bar, FIELD: psi);
- Pressure on equivalent radius (METRIC: bar, FIELD: psi) Pressure on equivalent radius is calculated the following way: for every connection the pressure on equivalent radius is calculated using the corresponding formula, then the sum for connections weighted by pore volume is calculated (formula for equivalent radius is in the section of UserManual Pressure equivalent radius calculation);
- Drawdown (METRIC: bar, FIELD: psi).

This graph in graphical interface is calculated as $WBP - WBHP$. WBHP – bottom hole pressure, WBP – average pressure in grid blocks containing connections adjusted to reference depth (density in the well bore is used) and using por volume weighted average (if something else is not set in keywords **WPAVE** (see 12.18.196), **WPAVEDEP**, see 12.18.197).

Drawdown in **WELDRAW** (see 12.18.109) is calculated differently: liquid or gas rate weighted average of productivity is used. Density in the well bore is not taken into account in AVG calculations, as the difference (in block and in connection) is taken at the same depth;

- WBP (METRIC: bar, FIELD: psi) (keywords **WPAVE** (see 12.18.196), **WPAVEDEP**, see 12.18.197);
- WBP4 (METRIC: bar, FIELD: psi) (keywords **WPAVE** (see 12.18.196), **WPAVEDEP**, see 12.18.197);
- WBP5 (METRIC: bar, FIELD: psi) (keywords **WPAVE** (see 12.18.196), **WPAVEDEP**, see 12.18.197);
- WBP9 (METRIC: bar, FIELD: psi) (keywords **WPAVE** (see 12.18.196), **WPAVEDEP**, see 12.18.197);

- Loaded pressure (METRIC: bar, FIELD: psi) (loading procedure is described in Section Load Well Graphs).

Pressure graphs for well I1 are shown in the figure 118.

For the Object Item **Well Connection** the following graphs are available:

- Connection pressure (METRIC: bar, FIELD: psi). The pressure inside a well bore

$$bhp + \rho \cdot g \cdot (ConnDepth - WellReferenceDepth)$$

where:

- *bhp* – bottom hole pressure;
- *ConnDepth* – connection depth;
- *WellReferenceDepth* – datum depth (specified by the keyword **WELSPECS**, see 12.18.3).
- Bulk pressure (METRIC: bar, FIELD: psi) (in the block with connection). This pressure is also visualized on the Calculated Map. Pressure;
- Drawdown (METRIC: bar, FIELD: psi);
- Connection Head Term (METRIC: bar, FIELD: psi) is calculated as a difference of well bottom hole pressure and a pressure in the wellbore in this connection.

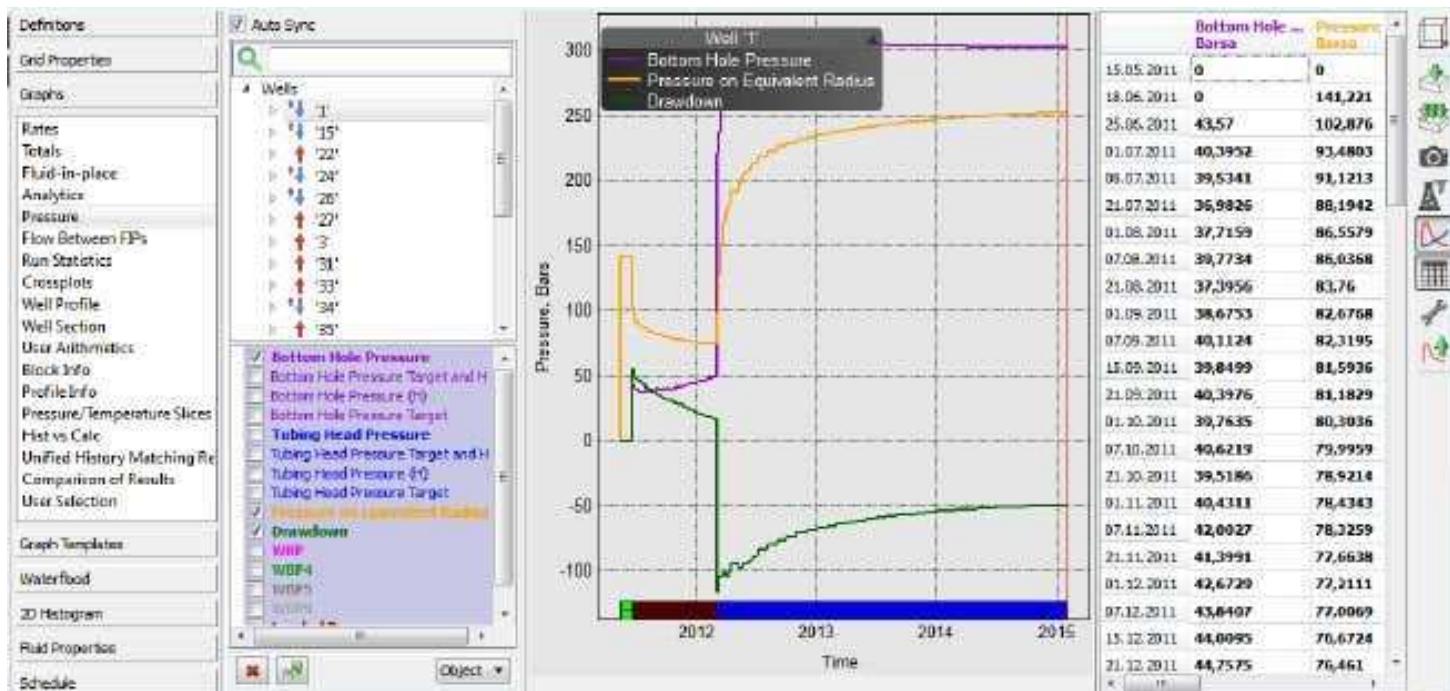


Figure 118. Well Pressure Graphs.

To visualise these graphs, you should select the well connection required in the Object Items dialog (on figure 119 the well connection (perforated interval) [18, 20, 6]) is selected. The figure shows graphs of pressure for the well connection and the block.

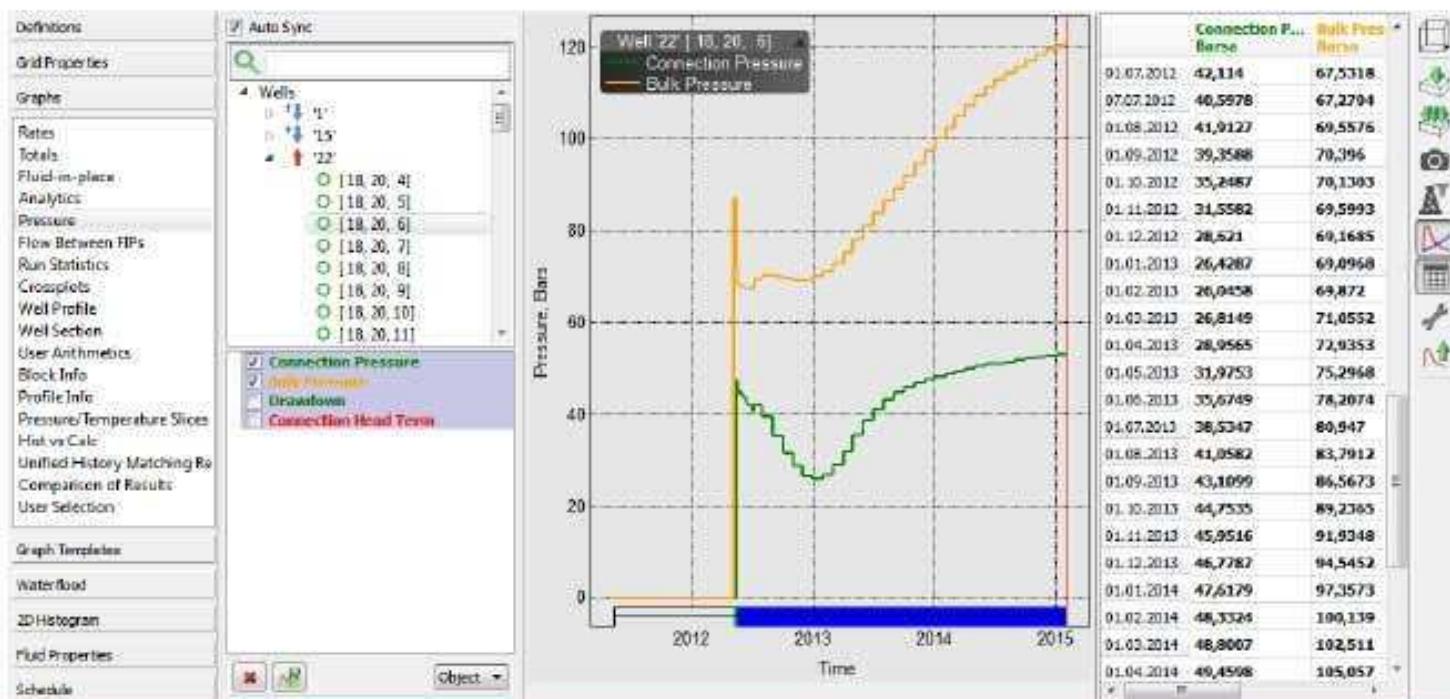


Figure 119. Well connection pressure graph.

For the Object Item **FIP Region** (FIPNUM), this tab will display the graphs listed below (figure 120):

- Avg. pressure (pressure weighted by hydrocarbons [whc]) (METRIC: bar, FIELD: psi);
- Phase potentials. The formulas for a calculation of potentials are described in the section Phase potentials calculations of tNav User Manual.
 - Avg. Gas Potential (calculated the way described in 31-parameter of **OPTIONS** (see 12.18.225) keyword) (METRIC: bar, FIELD: psi);
 - Avg. Oil Potential (calculated the way described in 31-parameter of **OPTIONS** (see 12.18.225) keyword) (METRIC: bar, FIELD: psi);
 - Avg. Water Potential (calculated the way described in 31-parameter of **OPTIONS** (see 12.18.225) keyword) (METRIC: bar, FIELD: psi);
- Avg. pressure [wpv] (pressure weighted by pore volume [wpv]) (METRIC: bar, FIELD: psi).

The pressure weighted by pore volume [wpv] is calculated as follows: the pressure in each block is multiplied by the pore volume of the block. Then the products of all the blocks are summed up, and the total is divided by the sum of all the blocks' pore volumes. The pressure weighted by hydrocarbons is calculated as follows: a pressure in each block is multiplied by the block's pore volume and by the sum of oil and gas content in the block. Then the products of all the blocks are summed up, and the total is divided by the sum of all the blocks' pore volumes multiplied by the sum of oil content and gas content of the block in question.

Segment object. A segment well structure allows to describe flow more accurately. A well is split in to parts – segments, then each segment has its own set of parameters. Links

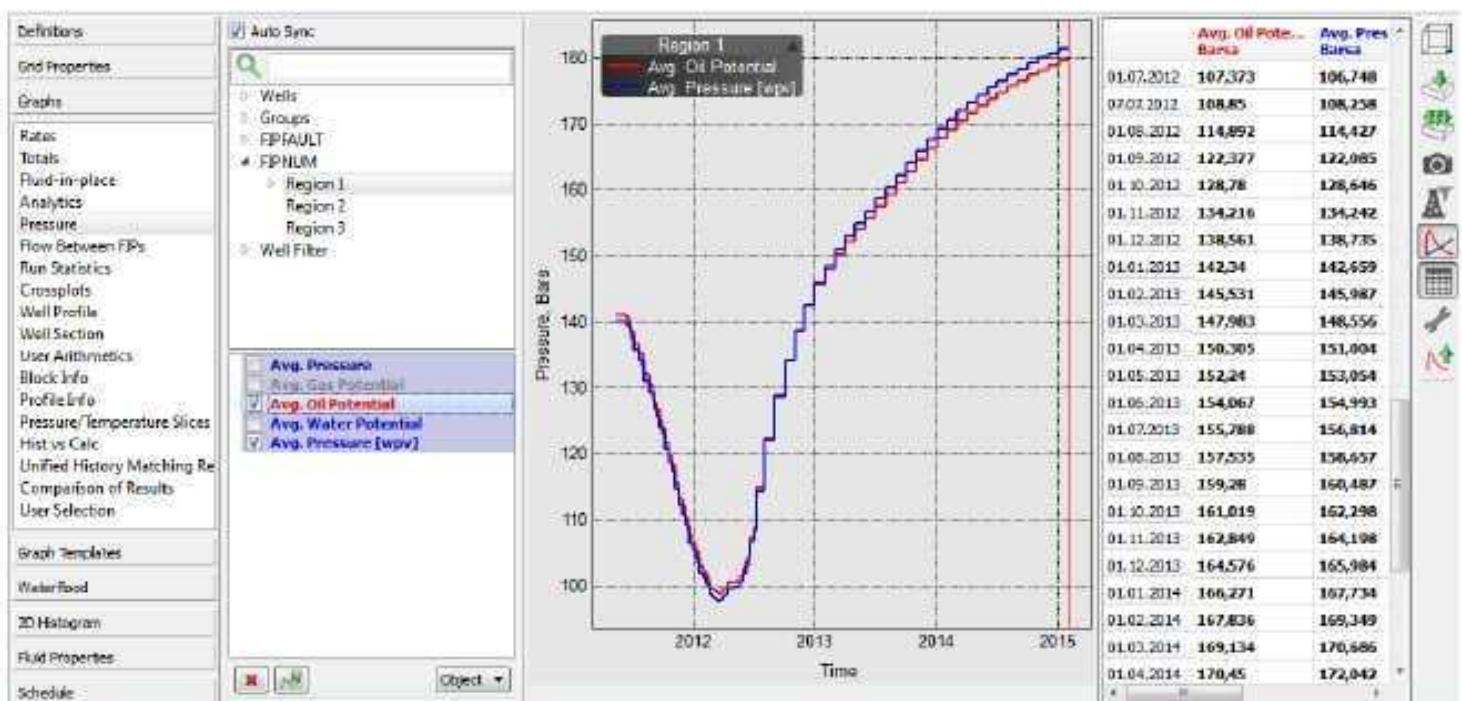


Figure 120. Graphs of pressure for the FIP region.

to the keywords which specify multisegment wells are in the section Multisegment well of tNav User Manual.

The main parameter – pressure drop in a segment. A pressure in each segment is equal to sum of pressure in a segment above and a pressure drop. There are three types of pressure lost: due to a hydrostatic, due to a friction and due to an acceleration of fluid.

The following graphs are available:

- pressure (METRIC: bar, FIELD: psi);
- pressure drop (METRIC: bar, FIELD: psi);
- pressure drop due to hydrostatic (METRIC: bar, FIELD: psi);
- pressure drop due to friction (METRIC: bar, FIELD: psi);
- pressure drop due to acceleration (METRIC: bar, FIELD: psi).

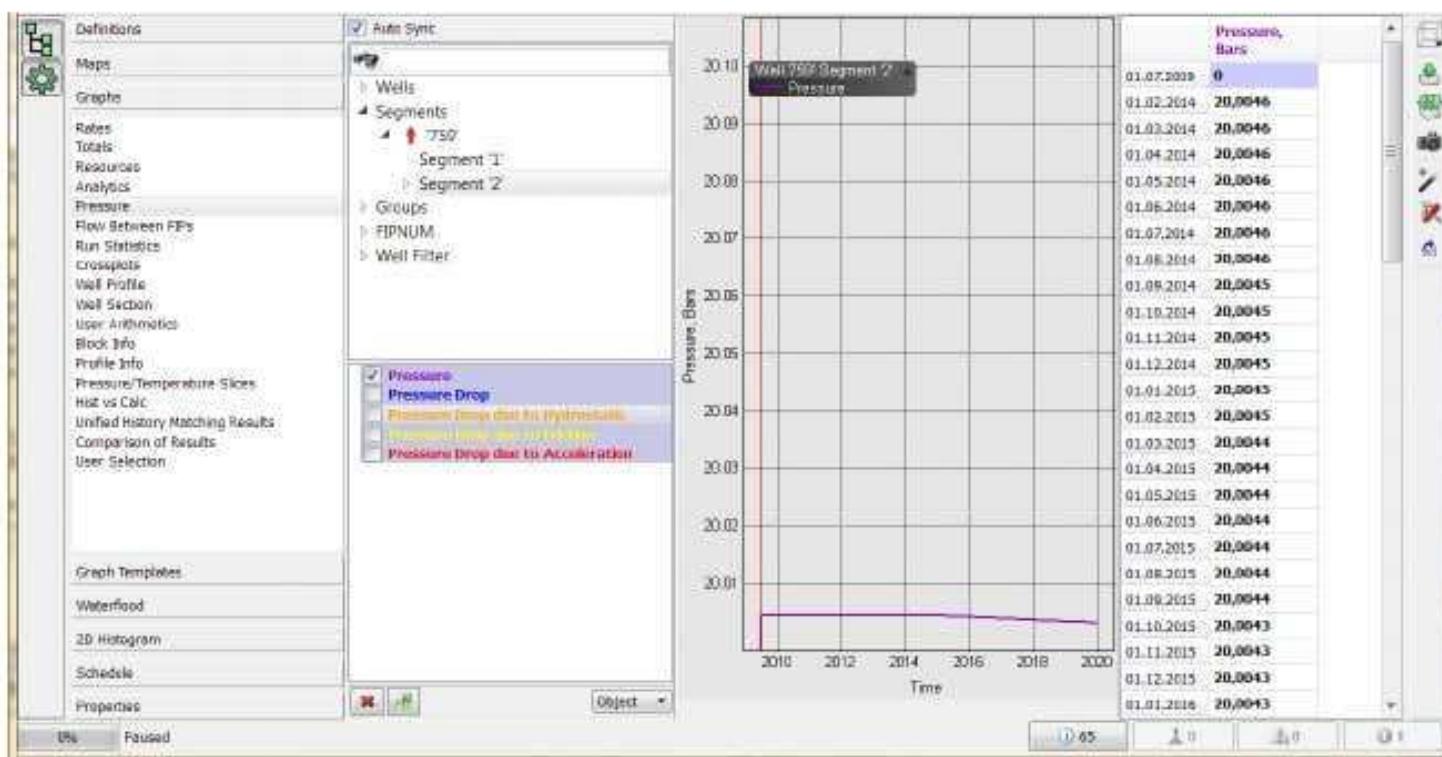


Figure 121. Segment's pressure.

Network object.

See the detailed description in the section NETWORK option. Automatic chokes. Compressors of tNavigator User Manual.

The following graphs are available:

- Node pressure (METRIC: bar, FIELD: psi).

In the figure 122 the pressure in network's node is shown.

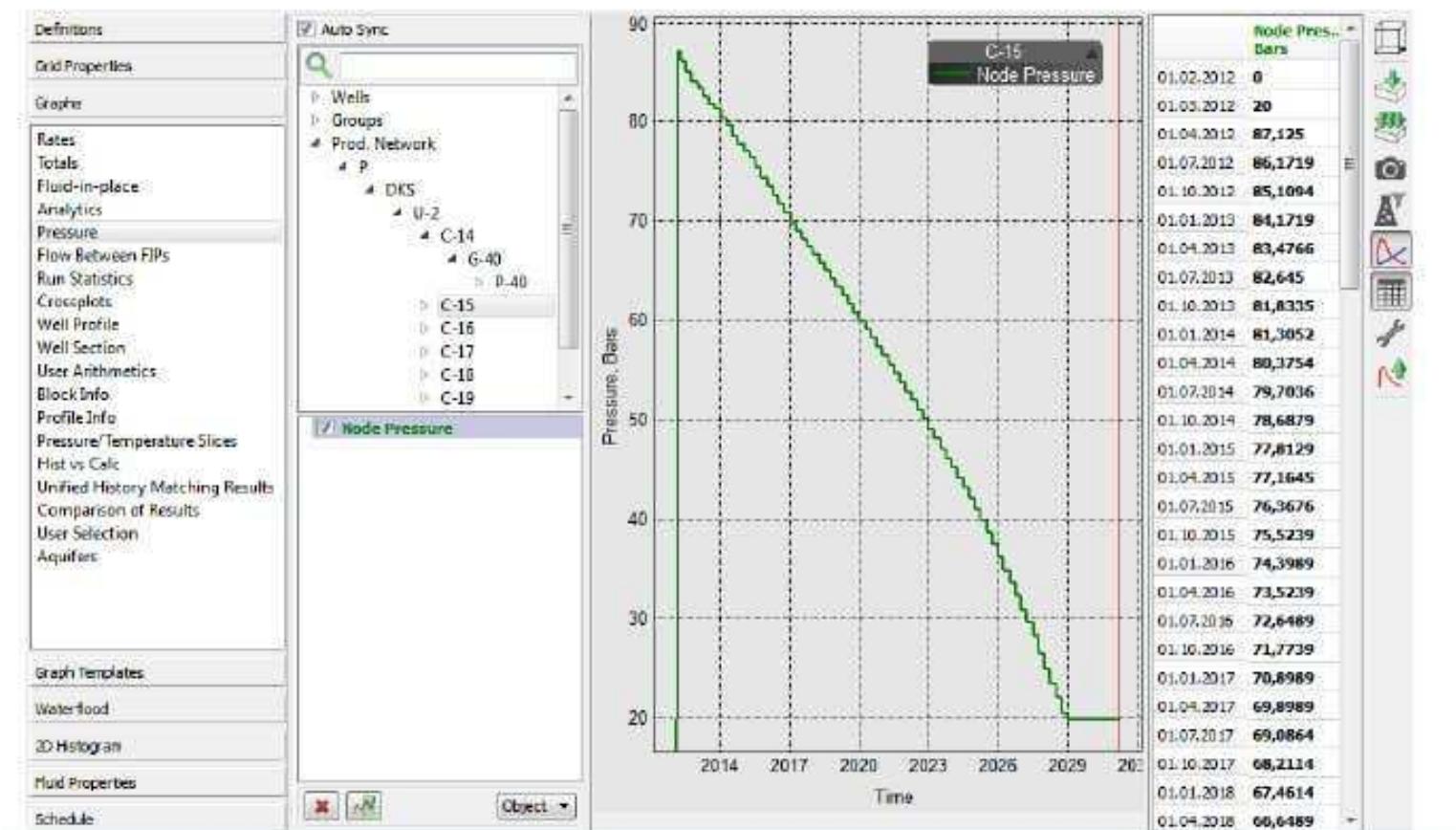


Figure 122. Node network pressure.

8.6. Flow Between FIPs

To view the detailed description of FIP Regions click on FIP Regions.

To visualize graphs of flow between FIPs, click the tab **Enable FIP Flow Calculation** before running a computation (in the right bottom corner) or set in advance the keyword **RPTMAPS** (see 12.15.51) parameter **FIPFLOW**.

The tab will visualize graphs of crossflows between FIP regions:

- water flow (METRIC: m³, FIELD: stb);
- oil flow (METRIC: m³, FIELD: stb);
- gas flow (METRIC: m³, FIELD: Mscf);
- component flow (METRIC: kg-mol; FIELD: lb-mol).

FIP region 3 and FIP region 2 are set as follows: **3 <-> 2**.

Figure 123 shows graphs of water, oil, and gas crossflows between the FIP region 3 and the FIP region 2 [3 <-> 2]. The data are also shown in the table on the right side.

For FIP regions, you can visualize several types of graphs in the tabs Rates, Totals, Analytics, Pressure. In the Object Items dialog, you should select the required FIP region (FIPNUM).

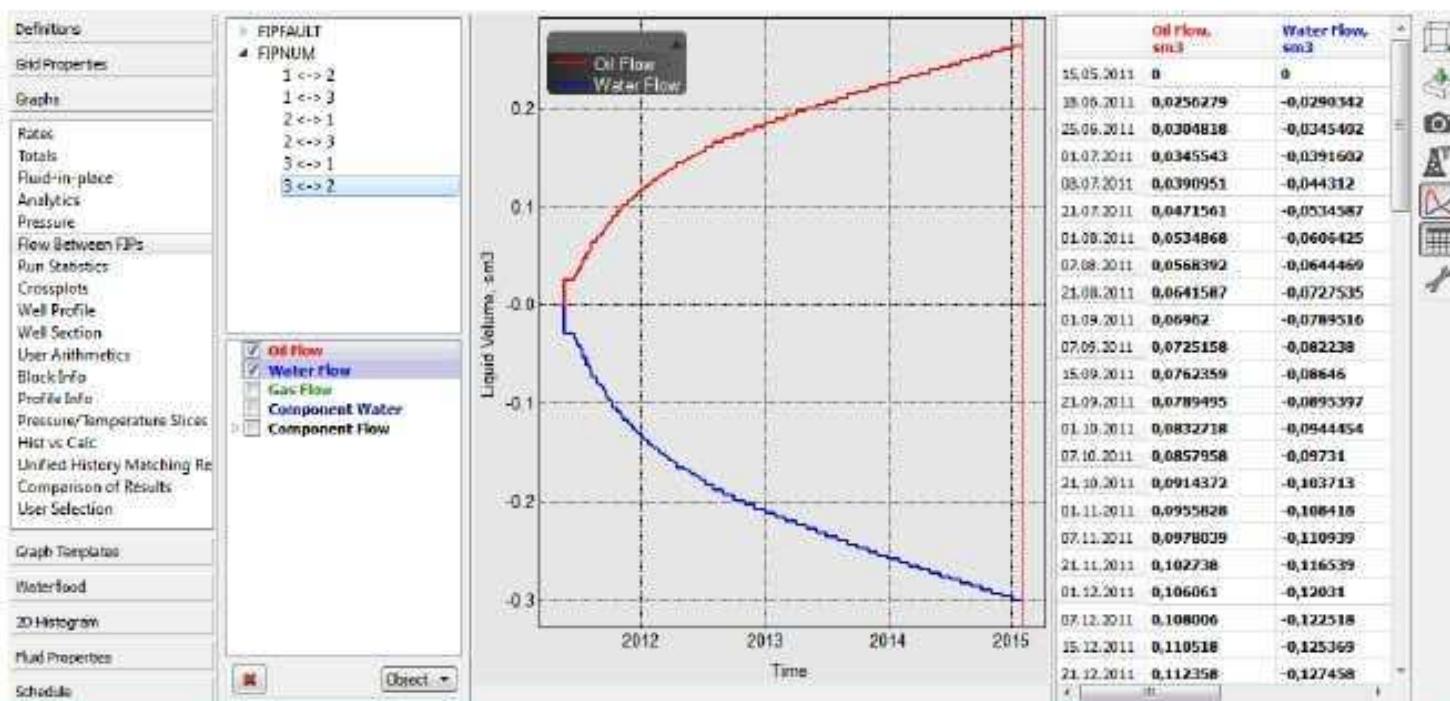


Figure 123. Crossflows between FIP regions.

8.7. Run Statistics

Run statistics contains the following graphs:

- Instant material balance error;
- Accumulative material balance error;
- DP (the maximum pressure change per time step in all the blocks);
- DN water (the maximum change of water's molar density per time step in all the blocks);
- DN oil (the maximum change of oil's molar density per time step in all the blocks);
- DN gas (the maximum change of gas's molar density per time step in all the blocks);
- DV (the maximum change of pore volume per time step in all the blocks);
- Maximum time step duration (days);
- Minimum time step duration (days);
- Average time step duration (days);
- Number of time steps;
- Number of Newton iterations;
- Number of linear iterations;

- Number of computational threads;
- Calculation time (the duration of each time step computation) (seconds);
- Total calculation time (from the start to now) (seconds);
- Total CPU time.

tNavigator run settings are set by the keyword **RUNCTRL** (see 12.18.124).

Run Statistics Graphs of current and cumulative material balance error are shown in the figure 124.

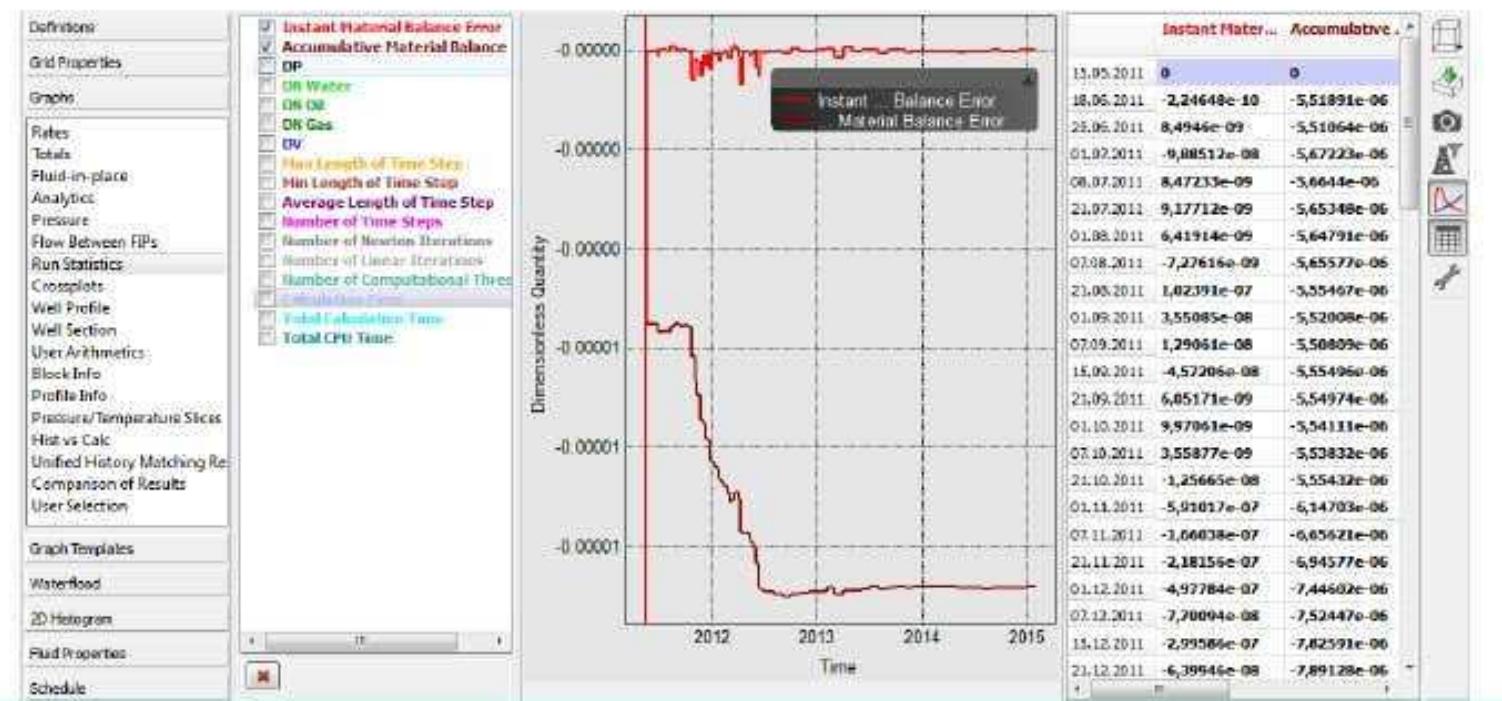


Figure 124. Run Statistics Graphs.

8.15. Historical vs. Calculated (Hist vs Calc)

This tab compares historical versus calculated production and injection parameters. The X-axis shows calculated values, the Y-axis shows historical values. There is a drop-down menu in which you can select one of the graphs listed below:

- Oil rate;
- Water rate;
- Gas rate;
- Liquid rate;
- Water injected rate;
- Gas injected rate;
- Oil total;
- Water total;
- Gas total;
- Liquid total;
- Water injected total;
- Gas injected total;
- Oil injected total;
- Tubing-head pressure;
- Bottom hole pressure.

Graphs can be visualized for **Wells** or **Groups** (of wells) (depending on your selection from the drop-down menu).

Legend.

The green line is the bisecting line of the angle. For wells on that line, historical values equal the calculated ones.

The red lines show deviations from the bisector (by default, the deviations should be at least 10 per cent (the tolerance level). Wells in the sector between the red lines (green squares) are considered to have an acceptable difference between the calculated parameter and the historical one. Wells outside that sector have major differences between the historical parameter and the calculated one (red squares).

Wells inside the **black rectangle** are medium-productivity wells (not classified as **key producers** – see below).

Wells inside the **second black rectangle** are low-productivity wells (with rates below the minimum rate set – see below). Initially, there is no second black rectangle.

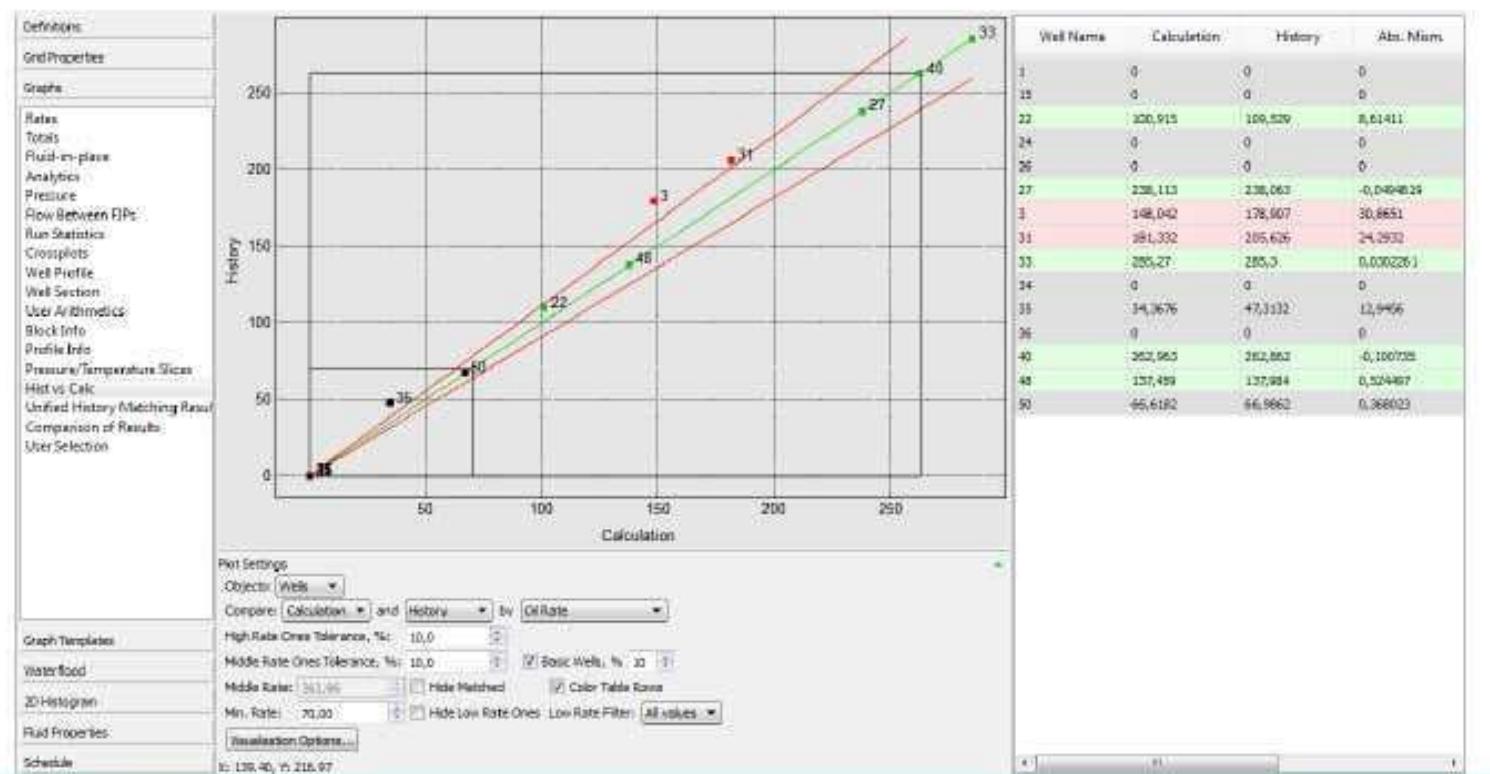


Figure 142. Calculated and Historical Graphs.

Tables on the right. The green rows show history-matched wells (inside the red-line sector), the red rows show non-history-matched wells (outside the red-line sector), the gray rows are low-productivity wells.

Graph Settings.

Tolerance. You can set acceptable deviations of calculated parameters from historical ones (tolerance level). The default deviation margin is 10 percent. The tolerance is set separately for high-productivity and middle-productivity wells. In the figure 142 the tolerance is set equal to 5% for high-productivity wells and 10% for medium-productivity wells. You can see the difference in the red lines inside and outside the black rectangle.

Basic wells - Key producers (the default setting is 10% of the total well count). The number of key producers is calculated as follows:

1. Sum up the calculated parameters of all the wells. For example, Oil total of well 1 is A_1 , Oil total of well 2 is A_2 . The sum for all wells is: $A_1 + A_2 + \dots + A_N = A$.
2. The number of key producers (B) is 10% of A . $B = 0.1 \cdot A$.
3. Take the wells in the decreasing order of the parameter in question (e.g. Oil total). Add Oil total parameters of well until you reach the volume of B . For example, you have summed up the values $A_3, A_{80}, A_{30}, A_{20}$ (Oil total values of wells 3, 80, 30 and 20 – in the descending order of Oil total).
4. The wells thus selected (well 3, 80, 30, and 20 in this example) are the key producers and are high-productivity wells. Other wells are classified as medium-productivity wells.

Middle rate. Calculate the average oil production rate for the medium-productivity wells.

Min rate. The default setting is zero. If you set a minimum higher than zero, all the wells with oil rates smaller than the minimum set will be classified as **low-productivity wells** (black squares) – see figure 142.

Hide matched. This will hide history-matched wells (the green squares between the red lines).

Hide low-rate ones. This will hide low-productivity wells.

Color table rows. Green rows are history-matched wells (between the red lines), red rows are wells not history-matched (outside the red lines).

Visualization options. You can set color, size and symbols to denote objects.

If you need to display only some of the wells (e.g., high-productivity wells only), create a Well Filter. Then only filter-selected wells will be shown.

8.16. Unified History Matching Results

This tab contains a history-match results table for all wells and parameters:

- Oil total;
- Water total;
- Gas total;
- Liquid total;
- Oil rate;
- Gas rate;
- Water rate;
- Liquid rate;
- Water injection rate;
- Gas injection rate
- Water-cut total;
- Water injection total;
- Gas injection total;
- THP – tubing-head pressure;
- BHP – bottom hole pressure.

For each parameter of each well, the table displays calculated and historical values, relative and absolute mismatches, and units are specified. The data are updated at every step of a computation. After the computation is completed, you can drag the slider to the required step and review the history-match table at that step.

Absolute Mismatch = Calculated Value – Historical Value.

$$\text{Relative Mismatch} = \frac{|\text{Calculated Value} - \text{Historical Value}|}{\text{Historical Value}} \cdot 100\%.$$

In the figure 143 a part of a summary history match table for Liquid Total is shown.

Well	Liquid Total			
	Calc., th. sm3	Hist., th. sm3	Rel. Res., %	Abs. Res., th. sm3
1260	16,9944	35,4907	52,1157	-18,4962
233	35,502	51,8808	31,5701	-16,3788
286	2,83357	23,0091	87,685	-20,1755
300	0,391419	0,521101	24,8861	-0,129682
301	3,24142	3,24142	0	0
302	4,57905	4,57905	0	0
303	0,414569	0,414569	0	0
304	0,042163	0,559417	92,463	-0,517254
305	2,66138	2,66138	0	0

Figure 143. Unified History-Match Table.

If you want the summary history-match table to show data for the required wells, set a Well Filter. The table will show wells covered by the filter.

You can sort data in the history-match table in the ascending or descending order of a parameter. To sort, click the title of the column to be sorted (Calc., Hist., Rel. Res., Abs. Res.). A triangle pointing up indicates that the data has been sorted ascending. If you want to change the sorting to descending, click the column's name again (you will see a triangle pointing down) (figure 144).

Columns of Unified History Matching Table can be moved around.

You can change the order of the columns using the  Preferences button on the right panel. By holding the left button on the mouse, you can drag the column titles and arrangement in the order preferred. This will change the arrangement of the columns in the table accordingly.

Well	Liquid Total			
	Calc., th. sm ³	Hist., th. sm ³	Rel. Res., %	Abs. Res., th. sm ³
311	0,160057	0,160057	0	0
310	0,115368	0,115368	0	0
306	4,44277	4,55624	2,49036	-0,113467
306B	4,66638	4,85086	3,80307	-0,184482
310B	1,25626	1,30677	3,86544	-0,0505125
316	2,1706	2,43847	10,9854	-0,267877
514B	132,97	155,929	14,7241	-22,9592
510	252,728	296,931	14,8866	-44,203
305B	143,633	168,948	14,9839	-25,3149

Figure 144. History-match table sorted in the order of increasing relative mismatch.

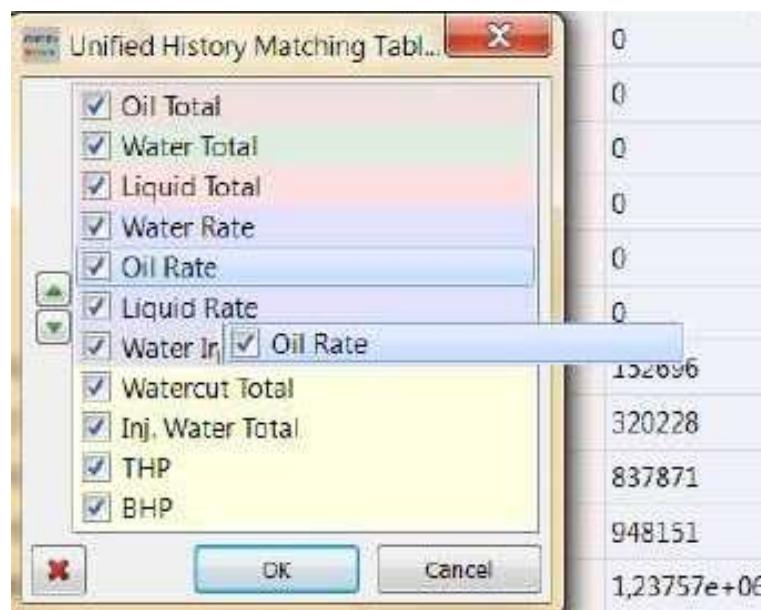


Figure 145. Changing the arrangement of columns in the history-match table.

Search well in the history-match table.

Click the button **Find a Well/conn**. Start typing the well name, and the wells with names matching with the typed symbols will be highlighted in blue and moved to the top of the list. The table's row with the selected well will be highlighted in blue – see figure 146.

Well	Oil Total				Water Total				Liquid Total		
	Calc., sm3	Hist., sm3	Rel. Res., %	Abs. Res., sm3	Calc., sm3	Hist., sm3	Rel. Res., %	Abs. Res., sm3	Calc., sm3	Hist., sm3	
I1	0	0	—	0	0	0	—	0	0	0	
I10	0	0	—	0	0	0	—	0	0	0	
I2	0	0	—	—	—	—	—	0	0	0	
I3	0	0	—	—	—	—	—	0	0	0	
I4	0	0	—	—	—	—	—	0	0	0	
I5	0	0	—	—	—	—	—	0	0	0	
I6	0	0	—	—	—	—	—	0	0	0	
I7	0	0	—	—	—	—	—	0	0	0	
I8	0	0	—	—	—	—	—	0	0	0	
I9	0	0	—	—	—	—	—	0	0	0	
P1	2,94237e+06	3,09506e+06	4,93353	—	—	—	—	—	—	—	
P10	3,42724e+06	3,69366e+06	7,21298	—	—	—	—	—	—	—	
P11	2,84523e+06	3,58606e+06	20,6584	—	—	—	—	—	—	—	
P12	2,67095e+06	2,30822e+06	15,7146	—	—	—	—	—	—	—	
P13	2,34023e+06	3,15758e+06	25,8852	—	—	—	—	—	—	—	
P14	2,46858e+06	2,98303e+06	17,246	-514455	1,01613e+06	501676	—	102,547	514455	3,48471e+06	

Figure 146. Search P1 well in the history-match table.

8.17. Comparison of Results

You can use this tab, if results from a different model have been loaded in this model. The table shows the main cumulative values of parameters for each model and their differences from the original one.

Use the button  **Settings** to select:

- Parameters:
 - Oil total;
 - Water total;
 - Liquid total;
 - Gas total;
 - Injected water total;
 - Average pressure.
- Differences from the base model:
 - Absolute values;
 - Relative values.
- Time steps:
 - All;
 - 1, 5, 15 years;
 - First and last steps;
 - Periodically (assign the period).

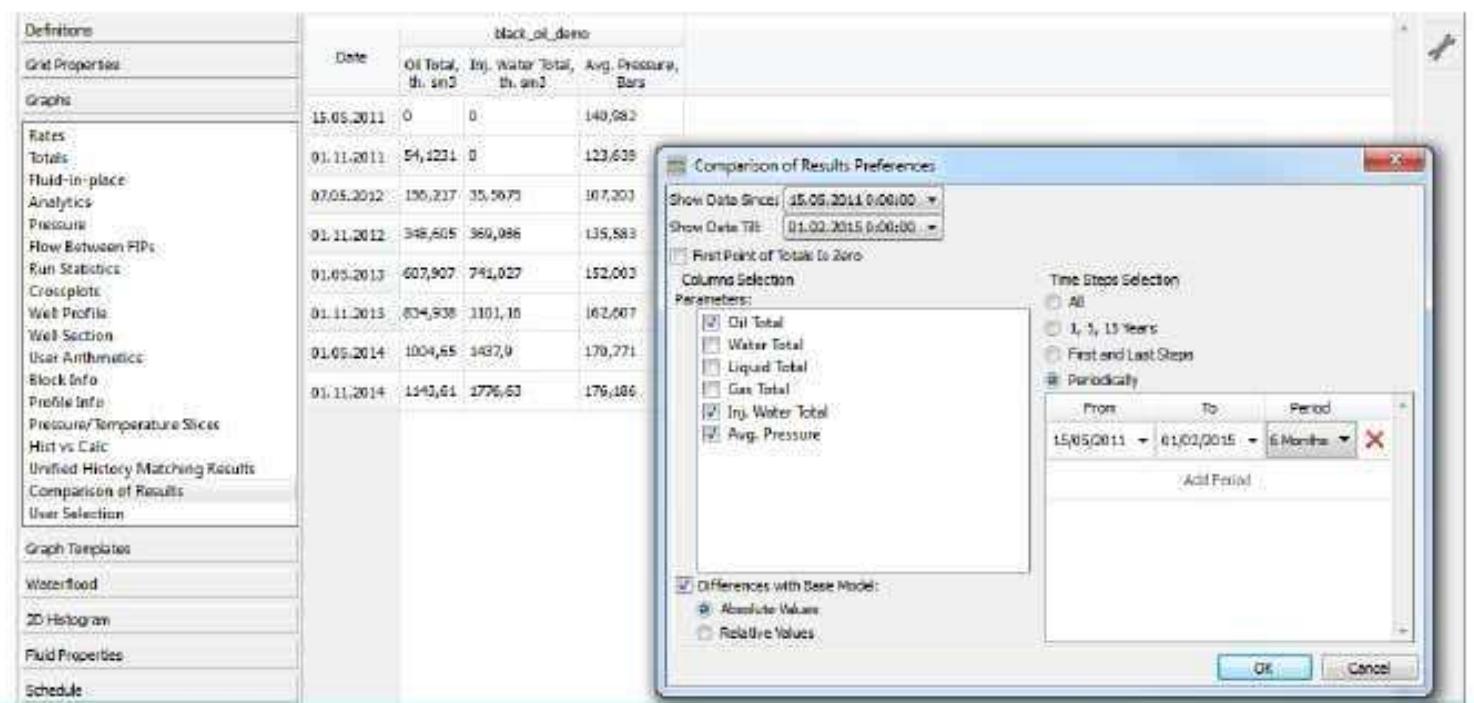


Figure 147. Comparison of results.

8.18. Well RFT Mismatch Table

This table shows the mismatch between calculated and measured RFT pressure data. This tab is enable if RFT (MDT) measurement were loaded to the model.

See the detailed description and examples in the training tutorial **8.1 How To Use RFT in History Matching**.

If zones (**ZONES**, see 12.4.28) are specified this data will be calculated for zones. If reservoirs are specified this data will be calculated for reservoirs as well (different zones can be combined into reservoirs in **ZONES** keyword).

8.19. Well PLT Oil Mismatch Table

This table shows the mismatch between calculated and measured PLT pressure data. This tab is available if PLT data are loaded to the model.

8.20. Tracers

Tracer graphs are only available if tracers are set for the model (with the keywords **TRACER** (see 12.7.1), **WTRACER** (see 12.18.154), **TRACERS** (see 12.1.44), **TBLK** (see 12.15.39), **TNUM** (see 12.15.40), **TVDP**, see 12.15.41), or if there are lumped pseudocomponents and their original components are monitored as tracers (keyword **LUMPING**, see 12.13.9), or if any of these is specified via GUI. For more information see training courses **2.2 How To Interactive Tracer Injection**, **2.3 How To Use Tracers Via Keywords**.

The parameters for each tracer:

- Tracer production rate (tracer production from this well during the time step) (METRIC: m^3/day , FIELD: stb/day);
- Tracer production concentration (tracer content in the fluid produced) (METRIC: kg/kg, FIELD: lb/lb);
- Tracer injection rate (tracer injection into this well during the time step) (METRIC: m^3/day , FIELD: stb/day);
- Tracer injection concentration (tracer content in the injection stream (METRIC: kg/kg, FIELD: lb/lb);
- Tracer production total (cumulative tracer production from this well) (METRIC: m^3 , FIELD: stb);
- Tracer injection total (cumulative tracer injection into this well) (METRIC: m^3 , FIELD: stb).

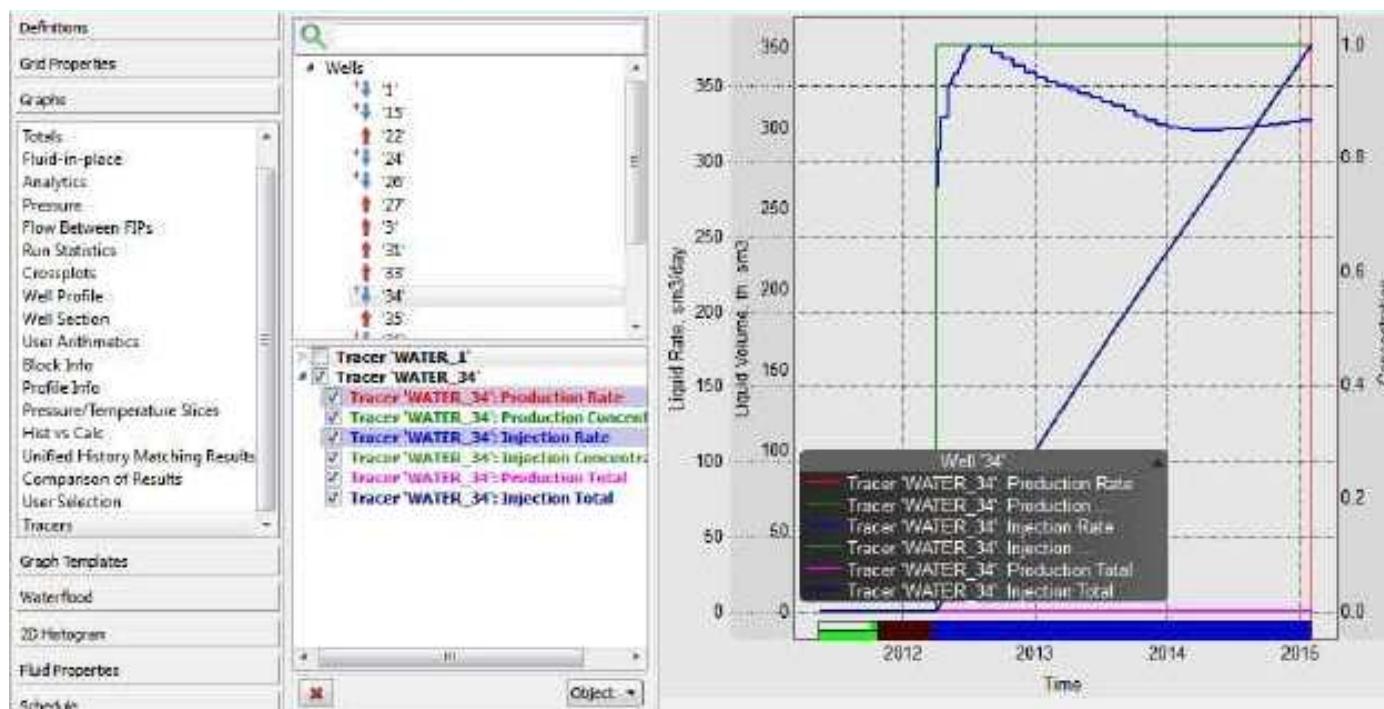


Figure 148. Tracer Graphs.

8.21. User selection

This tab contains user-selected graphs from other tabs (**Rates**, **Totals**, etc.). Initially the user selection is empty.

To add a graph to the **User Selection**, right-click the graph's title and select the feature **Add to User Selection** (see figure 149).

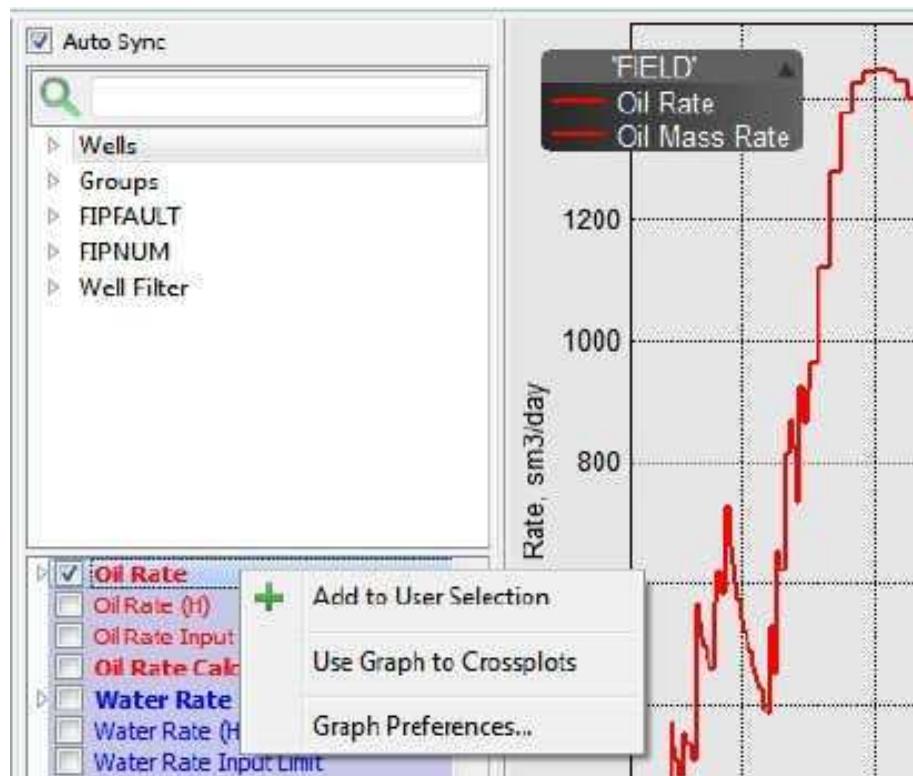


Figure 149. Adding a graph to the User Selection.

Go to the **User Selection** tab, where all previously selected graphs are. In the figure 150, there are three graphs in the User Selection: Oil rate, Water and Oil totals.

To remove a graph from the User Selection, right-click the graph's name and select **Remove** (figure 151). Selecting **Clear List** will remove all the graphs from the **User Selection**.

A User Selection will be saved for the model, when the model is closed and re-opened.

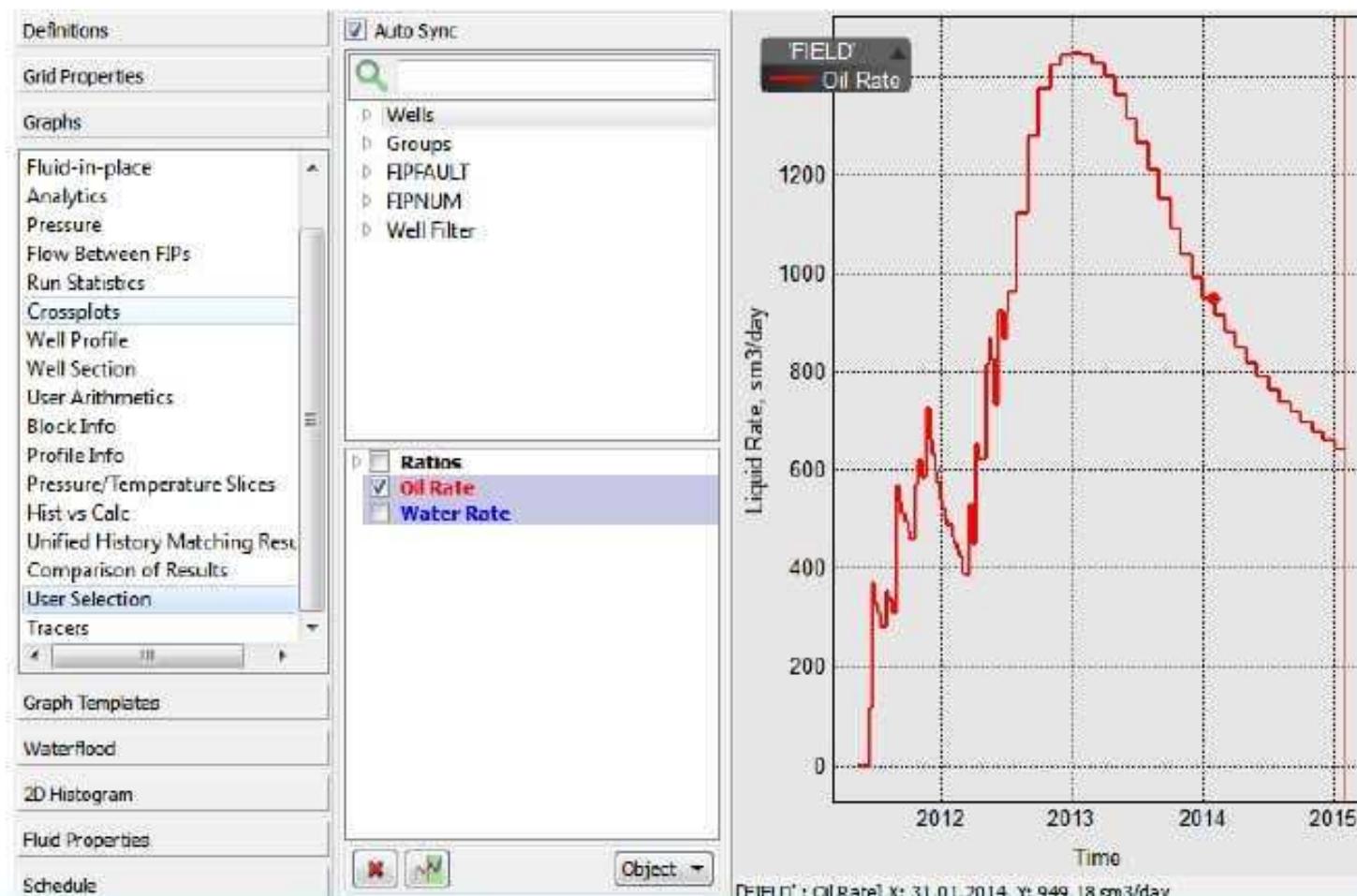


Figure 150. User selection.

8.22. Aquifers

See the detailed description in the section Inflow from aquifer of tNav User Manual.

This tab displays the graphs listed below for each aquifer (if there are no aquifers in the model, this tab will not be shown):

- Accumulative influx - cumulative water inflow (METRIC: m³, FIELD: ft³);
- Instant influx - step water inflow (METRIC: m³, FIELD: ft³);
- Instant influx rate - instantaneous water inflow (METRIC: m³, FIELD: ft³);
- Pressure (METRIC: bar, FIELD: psi).

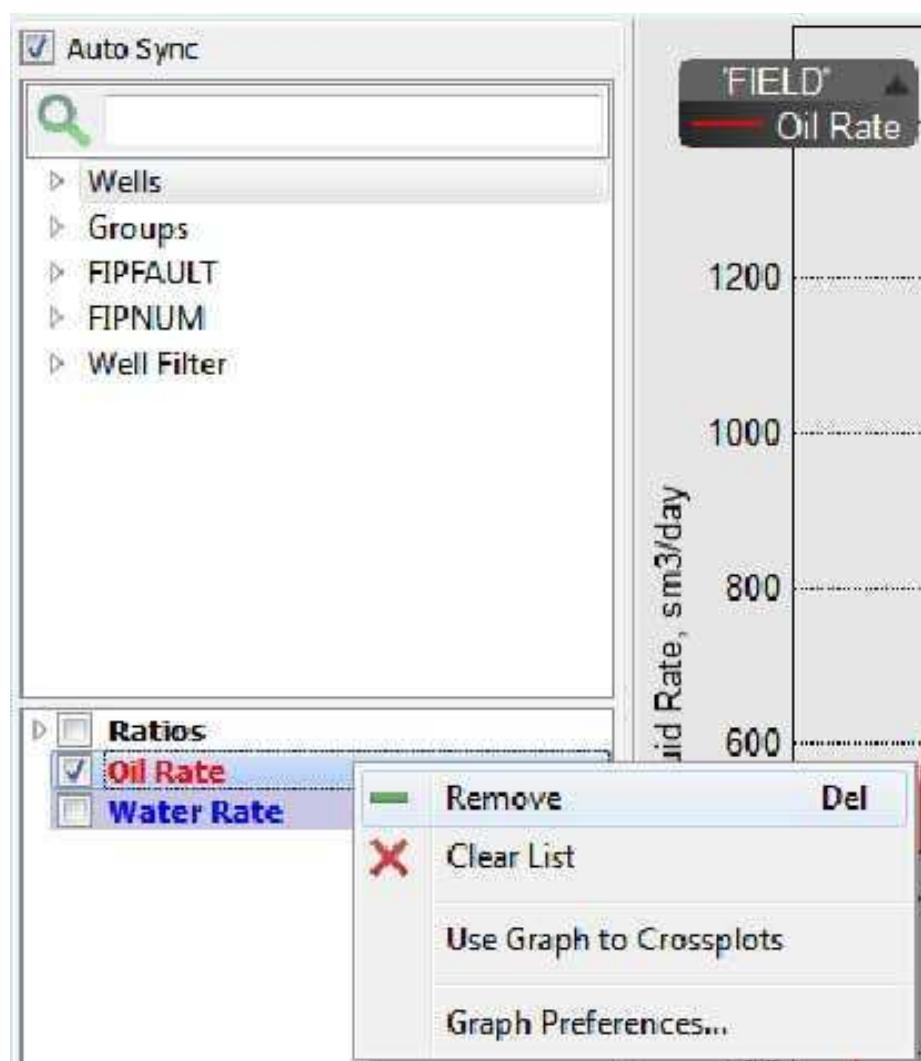


Figure 151. Removing graphs from the User Selection.

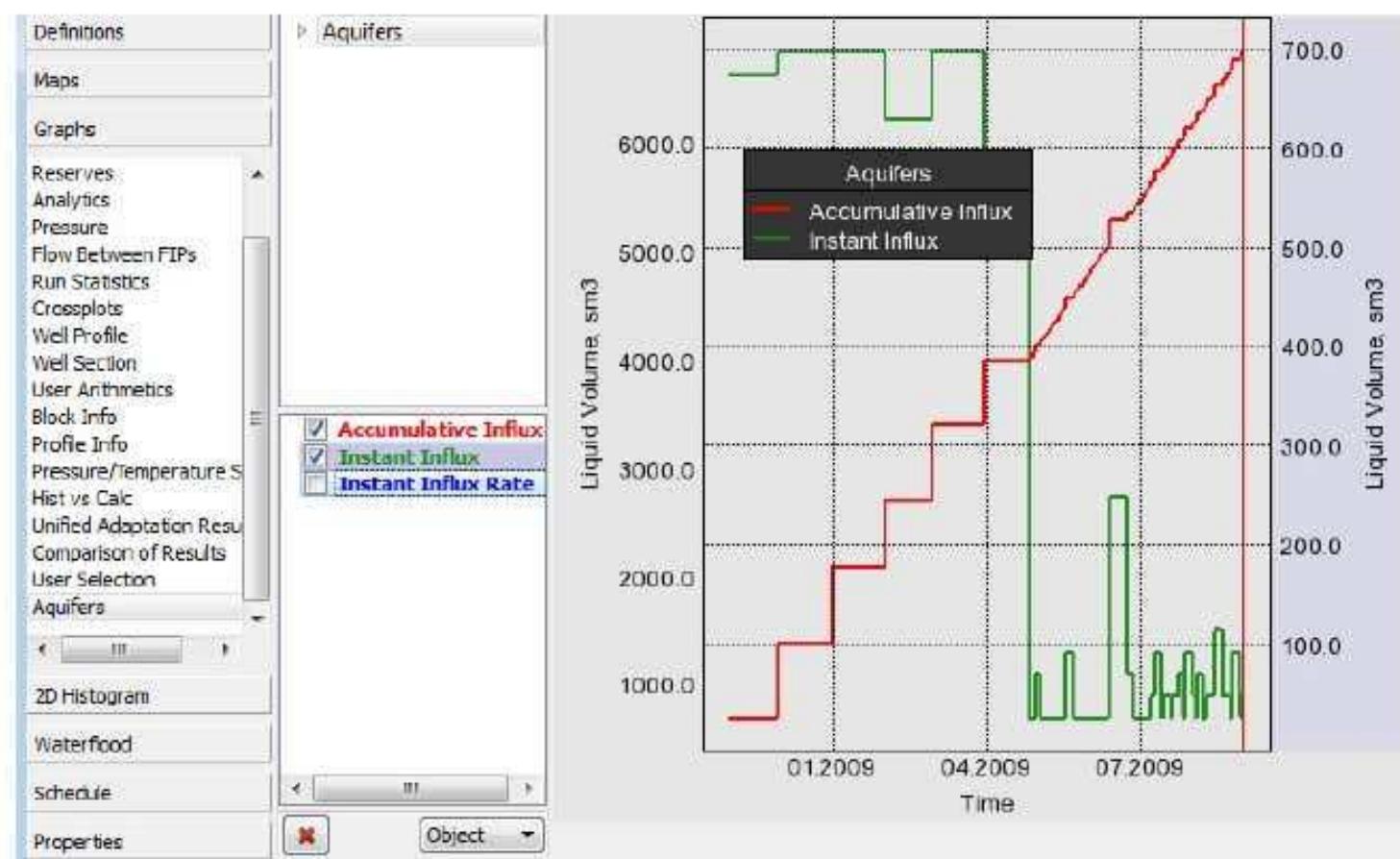


Figure 152. Aquifer.

9. Load Well Data

In this section there is a description of well data that can be loaded in graphical interface via the menu **Document. Load Well Data**.

Well data can be loaded into the model from the text files using **Schedule files dialog**: Layers, Trajectories, Groups, Events, History, History–FHF Format, Well Logs (LAS, RFT, PLT and other formats).

See examples in the following training tutorials:

- **3.1 How To Update Schedule;**
- **3.2 How To Add LAS Use Well Section;**
- **3.3 How To Load Well Data From Scratch;**
- **8.2 How To Use RFT in History Matching.**

9.1. Layers

File type: **Layers**.

File format – .txt.

Data description: layer name; z1-z2 (numbers along Z, to which this layer corresponds).

Example of this file format

```
'Layer_1'1-1  
'Layer_2'2-2  
'Layer_3'3-3  
'Layer_4'4-4
```

9.2. Trajectories

9.2.1. GWTD

File type: **GWTD**.

File format – .txt.

Data description: measured depth; x, y, z (negative).

Example of this file format

```
Well name: WELL1
3335.08379542 50133.99849282 57365.78811816 -3331.36235500
3350.53042953 50131.05636316 57365.30935266 -3346.51853724
3356.13983138 50129.97016088 57365.15669689 -3352.01963798
3364.20096452 50128.40792386 57364.95680241 -3359.92539399
```

9.2.2. Trajectory

File type: **Trajectory**.

File format – .dat, .txt.

Data description: well name; X; Y; Z (absolute depth); MD (depth along the well bore).

Example 1.

Example of this file format

```
welltrack 'WELL1'
100 110 2500.0      2500.000
100 110 2510        2510
100 110 2530        2540;
```

Example 2.

Load trajectory for multilateral well. First the data do the main branch goes. Then the data for additional branches goes. Branch number is set after two-spot sign.

Example of this file format

```
WELLTRACK 'WU20'  
3467.0031 -1259.4248 0 0  
3467.0031 -1259.4248 1430.9964 1430.9964  
3470.4462 -1260.7414 1443.0729 1443.6230  
3475.6100 -1262.7161 1454.4392 1456.2628  
3473.8894 -1262.0579 1471.3463 1473.2700 /  
  
/  
WELLTRACK 'WU20:1'  
3467.0031 -1259.4248 1430.9964 1430.9964  
3317.2285-1202.1550 1440.6577 1591.6377  
3124.4150 -1128.4284 1447.3353 1798.1739  
2967.7541 -1068.5256 1450.3189 1965.9234  
2795.5993 -1002.6983 1452.4500 2150.2466 /  
/
```

9.2.3. LAS

File type: **LAS**.

File format – .las.

Data description: Standard las-format (X, Y, absolute depth, measured depth). The order of the columns can be changed in the emerging dialogue.

Example of this file format

```

~Version Information
#-----
VERS.          1.2:
WRAP.          NO:

~Well Information
#-----
#MNAME.UNIT      DATA           INFORMATION
#-----  -----
STRT.M          10.00: Top Depth
STOP.M          2288.00: Bottom Depth
STEP.M          10.00: Increment
NULL.           -999.25: Null Value
UWI.            UNIQUE WELL ID: 3070010341
WELL.           Well: 107L
DATE.           Date: 15022009
COMP.            Company:
FLD.             FIELD NAME:
LOC.              LOCATION:
PROV.            Province:
SRVC.            Company:

~Other Information
#-----
~A
2500 100 110 2500
2510 100 110 2510
2540 100 110 2530

```

9.2.4. Generalized

File type: **Generalized**. File format – .dev.

Data description: Generalized GWTD format. Well names must begin with WELLNAME: (any letters size). The order of the columns can be changed in the emerging dialogue. Values Z are not negative. It is possible to check the box **Reverse Z** in the dialogue.

Example of this file format

```

WELLNAME: 'WELL1'
1.030384e+007    5733795    -135.7    -135.7
1.030384e+007    5733795    -110.7    -110.7
1.030384e+007    5733795    -85.7     -85.7
1.030384e+007    5733795    -60.7     -60.7
1.030384e+007    5733795    -35.7     -35.7
1.030384e+007    5733795    -10.7     -10.7

```

9.2.5. Dip-circle**File type: Dip-circle.**

File format – .trj.

Data description: measured depth, angle (between Z-axis and well vector), azimuth (angle between Y-axis and well vector in X direction). Well names must correspond to the file names.

Example of this file format

```

20    0.75 206.50
40    1.00 206.50
60    1.50 206.50
80    4.50 206.50
100   9.75 206.50
120   11.00 205.50
140   13.12 205.50
160   15.25 206.50

```

9.2.6. WellHead**File type: WellHead.**

WellHead file must be loaded if Dip-circle file is loaded.

File format – .txt.

Data description: well name, altitude z0 and wellhead coordinates (x0, y0). Columns and their order can be selected in the emerging dialogue.

Example of this file format

40R	57	33025.7	23427.2
3359	54.7	31384.3	20405.9
3405	57.3	30162.1	20212.8
3451	54.7	31386.8	20401.8
3452	61.5	30890.9	21500.9

9.3. Groups

9.3.1. Well – Group

File type: **Well – Group**.

File format – .txt.

Data description: well name; group to which this well belongs.

Example of this file format

```
' WELL1'  ' SAT-1'  
' WELL2'  ' SAT-1'  
' WELL3'  ' SAT-2'  
' WELL4'  ' SAT-2'
```

9.3.2. Group – Wells

File type: **Group – Wells**.

File format – .txt.

Data description: group name; wells which belong to this group.

Example of this file format

```
' GRUP1'  ' PROD1'  ' PROD2'  ' PROD8'  ' PROD9'  
' GRUP2'  ' INJ1'  ' INJ2'  ' INJ3'  ' INJ4'  ' INJ5'  
' GRUP3'  ' WPR1'  ' WPR9'  ' WPR17'  
' GRUP4'  ' WELSEGM3'
```

9.3.3. Group – Parent Group

File type: **Group – Parent Group**.

File format – .txt.

Data description: group name; parent group name.

Example of this file format

```
' GRUP1'  ' GRUP4'  
' GRUP2'  ' GRUP4'  
' GRUP3'  ' GRUP4'
```

9.4. Events

File type: **Events – Table**.

File format – .txt.

Data description: well name; branch, date; event; layer; lower depth; upper depth; radius; diameter; skin; multiplier.

Columns that are in the file should be selected in the drop-down menu. Order of boxes can be changed (in accordance with the data in the file).

Possible events:

- **perf** – open connections in all grid blocks where the trajectory intersects grid. Lower depth and upper depth should be specified;
- **sque** – shut connections in all grid blocks where the trajectory intersects grid. Lower depth and upper depth should be specified;
- **plug** – open connections in all grid blocks where the trajectory intersects grid. Upper depth should be specified, lower depth is calculated as the end of the trajectory;
- **bare** – shut connections in all grid blocks where the trajectory intersects grid. Upper depth should be specified, lower depth is calculated as the end of the trajectory.

Example 1.

Example of this file format

```
WELL1 1.7.1997 perforation 3354.8 3358.8 0.2 -3
WELL1 1.7.1997 perforation 3378.2 3381.6 0.2 -3
WELL1 1.7.1997 perforation 3383 3390.6 0.2 -3
WELL1 1.7.1997 perforation 3393.4 3394.2 0.2 -3
WELL1 1.7.1997 perforation 3397.5 3399.7 0.2 -3
```

Example 2.

Load perforations for multilateral wells. For the main well branch (first row) the default ranch number is used 1*; the next branch is set via number – 1. For each branch we set depth for perforated interval. Please choose **branch** for the corresponding column in the graphical interface.

Example of this file format

```
'WU20' 01.07.2012 1* 1440 1473 PERF 0.16
'WU20' 01.07.2012 1 1430 2150 PERF 0.16
```

Additional Settings.

- **Replace missing values with zero.** If this option is used, the parameters for the well that are missing in the file on the specific date will be replaced with zeros.
- **Data Filter.** If Data Filter is used, then historical data will be loaded only in the specified time period, including the First Date and the Last Date.

9.5. Well History

9.5.1. History table

File type: **Prod. history table**.

File format – .txt.

Data description: well name; date; oil rate; water rate; gas rate; liquid rate; gas injection; water injection; THP; BHP; well efficiency factor; polymer injection, enthalpy and other parameters.

Columns that are in the file should be selected in the drop-down menu. Order of boxes can be changed (in accordance with the data in the file).

Example 1. How loaded historical data is used.

Example of this file format

--Well	Date	WOPRH	WWPRH	WWIR
WELL15	01.10.2014	19.6224	130.378	0
WELL15	01.11.2014	19.1517	130.848	0
WELL15	01.12.2014	18.7443	131.256	0

In this example we load historical data for WELL15: oil rate (column WOPRH), water rate (column WWPRH) and water injection rate (column WWIR).

Loading the data in this format we consider that the well works in the following way:

- Oil rate $19.6224 \text{ sm}^3/\text{day}$ and Water rate $130.378 \text{ sm}^3/\text{day}$, from **01.10.2014 to 01.11.2014**;
- Oil rate $19.1517 \text{ sm}^3/\text{day}$ and Water rate $130.848 \text{ sm}^3/\text{day}$, from **01.11.2014 to 01.12.2014**;
- Oil rate $18.7443 \text{ sm}^3/\text{day}$ and Water rate $131.256 \text{ sm}^3/\text{day}$, from the date **01.12.2014**.

Two scenarios are possible:

- If the last date in the model is 01.12.2014, then the rates from the last line **are not taken into consideration** in cumulative production calculation. Oil cumulative production from 01.10.2014 is calculated as $19.6224 * 31 + 19.1517 * 30$ (only October+November). **To take December into account add the last date 01.01.2015 to the model (DATES, see 12.18.110).**
- If the last date in the model is 01.01.2015, then the rates from the last line **are taken into consideration** in cumulative production calculation. Oil cumulative production from 01.10.2014 is calculated as $19.6224 * 31 + 19.1517 * 30 + 18.7443 * 31$ (October+November+December).

Note. In graphical interface on the Graphs tab in the table on the right rates are visualized with date shift, see the picture 153.

2. Between yellow and green lines ($M - D \cdot \text{Sigma}-$, M).

A water injection will be balanced for wells within the sectors, wells outside the sectors will be disregarded. By default: $\text{Sigma}- = 1$, $\text{Sigma}+ = 1$, the balancing region is limited by the blue line and the yellow line. To change the balancing region, you need to change the values of **Sigma-** and **Sigma+**.

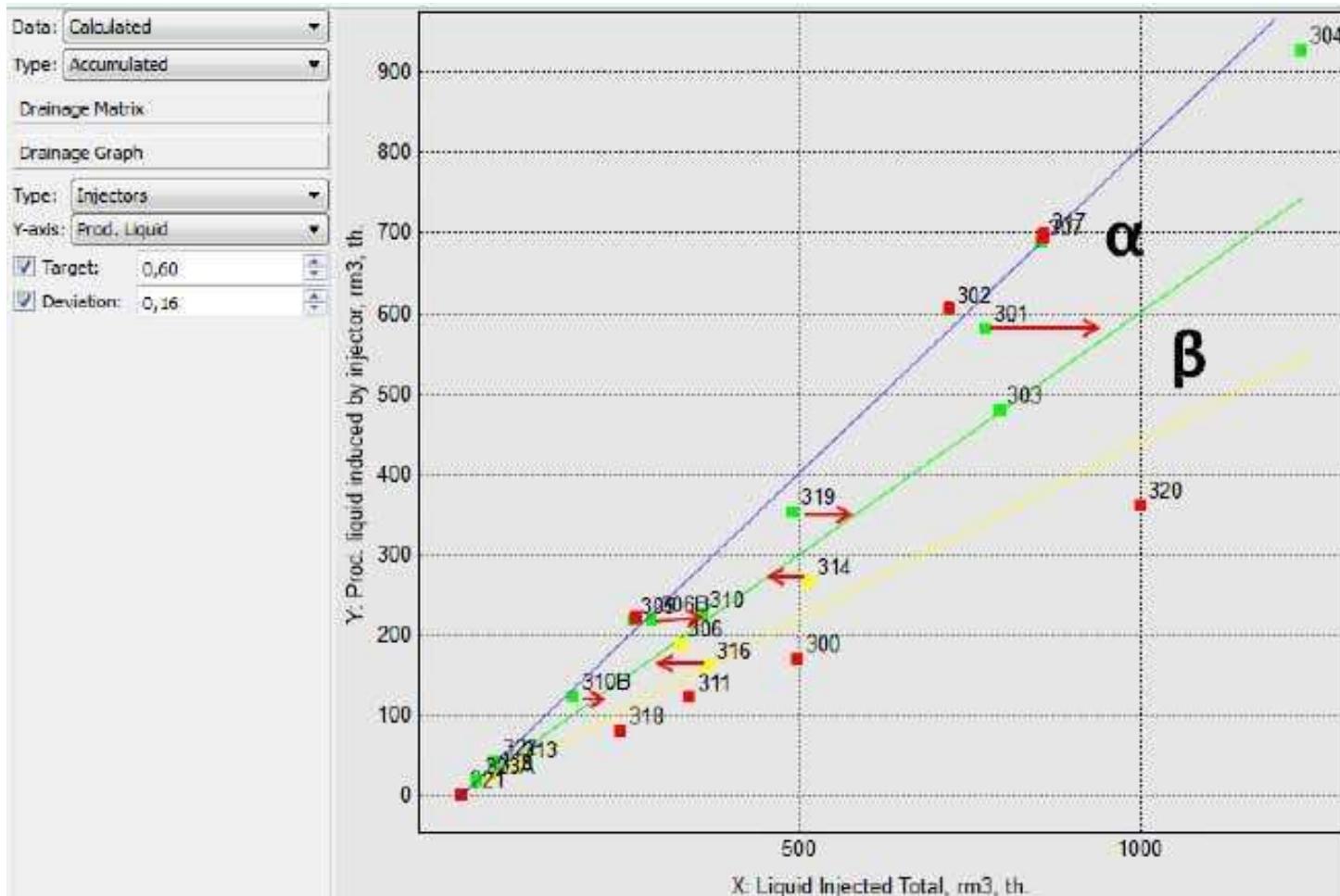


Figure 165. Balancing regions.

Plotting the Lines is on the 165.

For each injector i , the graph computes the ratio $\gamma[i]$ of liquid production to water injection. M is the average of all the $\gamma[i]$ values (the green line), D is the rms of all the $\gamma[i]$ values (yellow and blue lines).

Notice. The blue line is not visualized in the tNavigator graphical interface. In the figure 165 the blue line is added to show balancing regions.

You can modify the positions of yellow (**Deviation**) and green lines (**Target**) by checking the relevant box and typing the new value. But this modification will not affect the Balancing Region for water optimization.

The idea of Balancing (165):

1. A water injection will only be balanced for wells within the sectors, wells outside the sectors will be disregarded.

2. For any well within sector $(M, M + D \cdot Sigma +)$ water injection will be increased so as to bring the well closer to the green mean line.
3. For any well within sector $(M - D \cdot Sigma -, M)$, a water injection will be reduced so as to bring the well closer to the green mean line.

Balancing Formula.

In every subsequent step, a water injection volume will be multiplied by the following WEFAC (see 12.18.70) for each i -well:

- For wells in the sector $(M, M + D \cdot Sigma +)$ with *Alpha* multiplier between green and blue lines:

$$WEFAC_i = 1 + Epsilon + \left(\frac{Alpha \cdot (\gamma[i] - M)}{max\gamma[i] - M} \right)^{Degree}$$

- For wells in the sector $(M - D \cdot Sigma -, M)$ with *Beta* multiplier between green and yellow lines:

$$WEFAC_i = 1 + Epsilon + \left(\frac{Beta \cdot (M - \gamma[i])}{M - min\gamma[i]} \right)^{Degree}$$

An epsilon is used to increase a water injection volume.

Recommendations.

The value of parameter *Alpha* should not be large (this can cause a large watercut of producers corresponding to injectors in the sector $(M, M + D \cdot Sigma +)$). Parameter *Beta* can be large to make the injectors in the sector $(M - D \cdot Sigma -, M)$ inject water a little.

Parameter *Comp* is used for compensation. If *Comp* is not zero, then:

- For wells in the sector $(M, M + D \cdot Sigma +)$ with *Alpha* multiplier between green and blue lines:

$$WEFAC_i = 1 + Epsilon + Form \cdot \left(\frac{Alpha \cdot (\gamma[i] - M)}{max\gamma[i] - M} \right)^{Degree}$$

- For wells in the sector $(M - D \cdot Sigma -, M)$ with *Beta* multiplier between green and yellow lines:

$$WEFAC_i = 1 + Epsilon + Form \cdot \left(\frac{Beta \cdot (M - \gamma[i])}{M - min\gamma[i]} \right)^{Degree}$$

where $Form = \frac{\Delta Q}{Q} \cdot Comp$, ΔQ is the change of water injection volume in the current step, Q is the total water injection volume in the current step.

10.5. Waterflood compensation

You can use tNavigator to optimize water injection at each time step.

To optimize water injection through compensation of average reservoir pressure:

1. Select **Waterflood**;
2. Check **Drainage Matrix**;
3. Check **Compensation**;
4. In the pop-down menu, select **Average Reservoir Pressure**;
5. Run a computation and wait for it to be completed.
6. The modified injection parameters will be saved to the User file as follows. In the model folder, a USER sub-folder will be created (see details about USER subfolder in tNavigator User Manual) to which the file with the modified well schedule will be saved (see keyword **COMPENSATION**, see 12.18.106).

Compensation of Average Reservoir Pressure.

Set the value of **Multiplier**. In every subsequent step, injection will be computed as follows:

- Injection volume (under reservoir conditions) = **Production volume (under reservoir conditions)** × **Multiplier**.
- If the **Multiplier** is 1, the production volume under reservoir conditions will equal the injection volume under reservoir conditions (as can be seen in graphs of fluid production and injection under reservoir conditions). The average reservoir pressure shows insignificant changes (as seen in the pressure graph).

Injection is uniformly distributed among the wells, this option will not work with group control.

11. 2D Histogram. Crossplot

Using the tab **2D Histogram** it is possible to visualize:

- 2D Histogram;
- X Histogram;
- Y Histogram;
- Crossplot.

Using the simplest one-dimensional histogram (see figure 166) it is easy to understand the number of blocks corresponding to the high and low porosity. For example, as can be seen in figure 166 for the considered field the most part of the blocks has a porosity around 0.1.

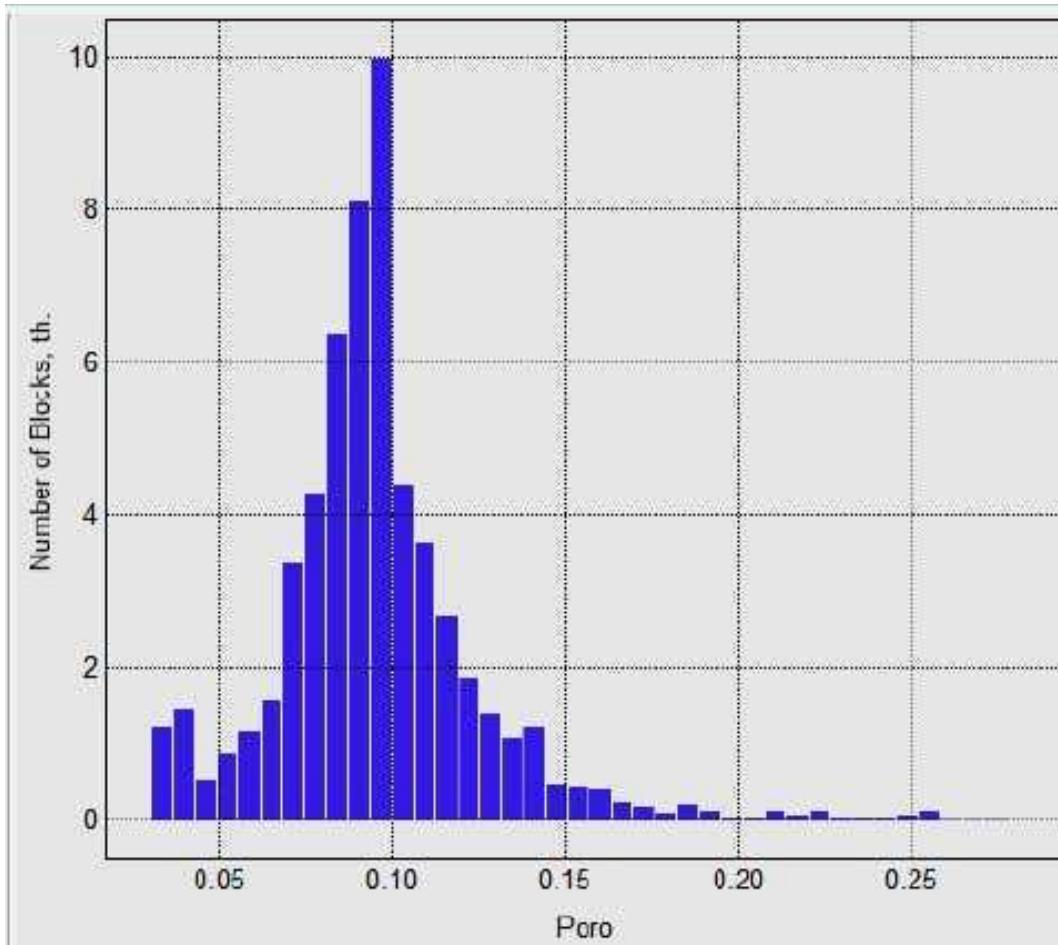


Figure 166. One-dimensional histogram.

11.1. 2D Histogram

A two-dimensional (2D) histogram is a method of color visualization of a distribution of values of arbitrary two functions through grid blocks. Here is one simple example.

A 2D histogram shows how many higher-porosity blocks (or low-porosity blocks) have high permeability and how many of them have a low-permeability – see figure 167.

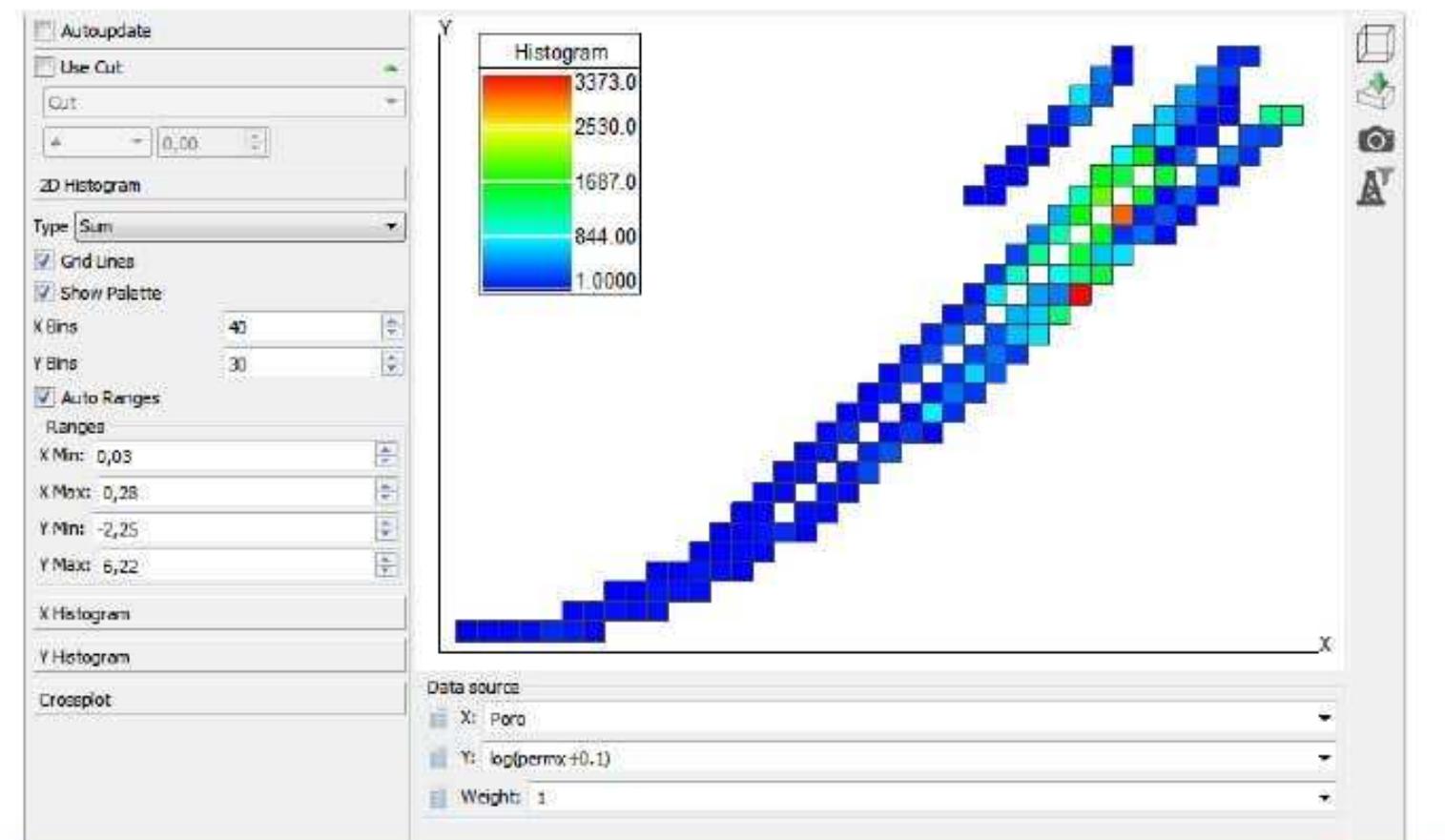


Figure 167. 2D histogram.

Creating a 2D Histogram.

Two selected functions (e.g., $f_1 = \text{poro}$ (porosity), $f_2 = \log(\text{perm}x + 0.1)$ (log of X-axis permeability)) are shown along two axes - X and Y. The value of each of these functions is calculated for each block. Then the X and Y axes are divided into sections:

$$[X_{min}, X_{min} + dx, X_{min} + 2dx, \dots, X_{max}] = [X_1, X_2, \dots, X_{Xbins+1}];$$

$$[Y_{min}, Y_{min} + dy, Y_{min} + 2dy, \dots, Y_{max}] = [Y_1, Y_2, \dots, Y_{Ybins+1}].$$

X_{min} and Y_{min} are the minimum values of the f_1 and f_2 functions in the grid blocks; X_{max} and Y_{max} are the maximum values of the functions in the grid. The number of sections (X_{bins} , Y_{bins}) can be adjusted (the number is set in **X bins** or **Y bins** fields).

For each bin of the X-axis $[X_i, X_{i+1}]$ and for each bin of the Y-axis $[Y_j, Y_{j+1}]$ the number of blocks having the values f_1 , f_2 in this range (i.e. $X_i \leq f_1 < X_{i+1}$, $Y_i \leq f_2 < Y_{i+1}$). When you put the cursor on a histogram cell, you will see the range and the number of blocks in the status bar. For example, in the model in the figure 167 752 blocks have porosities in the range [0.106, 0.113] and permeabilities in the range [1.99, 2.27]. So for each square $[X_i, X_{i+1}] \times [Y_j, Y_{j+1}]$, the blocks of the original grid with the function values within this range have a certain color in the 2D histogram.

It can be seen that blocks with the lowest permeabilities (dark-blue on figure 167) can have different values of porosity, for blocks with average permeabilities a porosity is related to the log of permeability. Low-permeability blocks have a low-porosity; high-permeability blocks (in the top-left part on figure 167) have a high-porosity.

When you activate a User Cut (check **Use Cut**), blocks not included in the filter will be

disregarded in the histogram.

You can also use additional parameters, such as **Weight** and **Type**.

Weight.

If you specify the **Weight** different from 1, for each square $X_{i+1}] \times [Y_j, Y_{j+1}]$ the model will sum up the weights – and not the number – of the blocks in which $X_i \leq f_1 < X_{i+1}$, $Y_i \leq f_2 < Y_{i+1}$. For example, if you select a weight that is the function $f_3 = \text{Soil} * \text{porv}$, the model will sum up the value of f_3 (the volume of oil in the reservoir) for all the blocks in which $X_i \leq f_1 < X_{i+1}$, $Y_i \leq f_2 < Y_{i+1}$.

Type.

The **Type** of histogram indicates how the blocks within the range to be handled. If the type is **Sum**, the blocks' weights will be summed up. If the type **Average** is selected, the model will calculate the average weight of the blocks within the range. If the type is **RMS**, the model will calculate the RMS of the weights of the blocks within the range.

Another 2D Histogram Example.

Input boxes at the bottom of the image: $\tilde{\mathbf{O}}$ is the function reviewed along the X-axis (designated f_1). \mathbf{Y} is the function reviewed along the Y-axis (designated f_2). Weight is the weight function for histogram computations (f_3).

If on figure 168:

- $f_1 = I$;
- $f_2 = J$;
- $f_3 = \text{Soil}$.

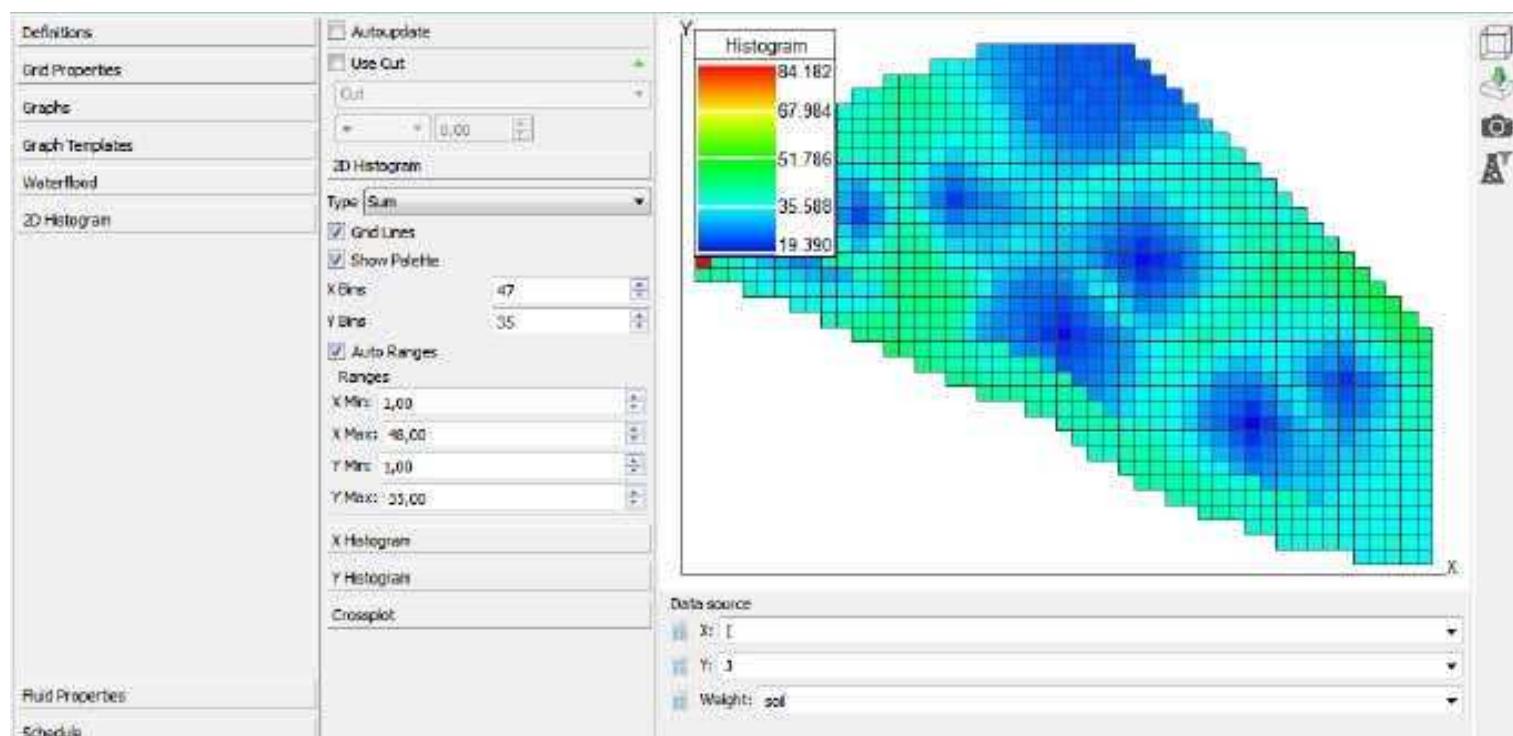


Figure 168. Weighted 2D histogram.

Parameter values (the right panel):

X bins – the number of bins into which the X-axis is split (Xbins). **Y bins** – the number of bins into which the Y-axis is split (Ybins).

The default values are equal (40, 30). In this example, the number of bins equals the number of blocks in the model.

Range.

If you check **Auto Min-Max**, the boundaries of the square will be determined automatically as the minimum value and the maximum value of the functions set along the X-axis and the Y-axis (the functions will be calculated for each grid block). X_{min} = the minimum value of f_1 , X_{max} = the maximum value of f_1 , Y_{min} = the minimum value of f_2 , Y_{max} = the maximum value of f_2 . If you uncheck this feature, you can set your own minimum and maximum values.

Type. The Type options are:

- Sum;
- Average;
- RMS.

The histogram will only use data of the grid blocks covered by the filter, with the f_1 and f_2 values within the following ranges: $X_{min} \leq f_1 \leq X_{max}$, $Y_{min} \leq f_2 \leq Y_{max}$.

The X-axis and the Y-axis are split into bins with the lengths of dx and dy, where:

- $dx = \frac{X_{max}-X_{min}}{X_{bins}}$;
- $dy = \frac{Y_{max}-Y_{min}}{Y_{bins}}$.

So the X-axis and the Y-axis are divided as follows:

$$\begin{aligned}[X_{min}, X_{min} + dx, X_{min} + 2dx, \dots, X_{max}] &= [X_1, X_2, \dots, X_{X_{bins}+1}]; \\ [Y_{min}, Y_{min} + dy, Y_{min} + 2dy, \dots, Y_{max}] &= [Y_1, Y_2, \dots, Y_{Y_{bins}+1}].\end{aligned}$$

For this axis split, 2D histogram grid (as opposed to the model grid) is created. The 2D histogram grid, the block of $[X_{i+1}] \times [Y_j, Y_{j+1}]$ will display a value that will be, depending on the histogram type:

- A sum.

The value will be equal to the sum of the values of the weighted function f_3 in the grid blocks for which $X_i \leq f_1 < X_{i+1}$, $Y_i \leq f_2 < Y_{i+1}$;

- An average.

The value will equal the average value of the weighted function f_3 of the grid blocks for which $X_i \leq f_1 < X_{i+1}$, $Y_i \leq f_2 < Y_{i+1}$.

- An rms.

The value will equal the rms of the weighted function f_3 of the grid blocks for which $X_i \leq f_1 < X_{i+1}$, $Y_i \leq f_2 < Y_{i+1}$.

The values generated will be color-designated in the histogram.

2D Histogram's Matches with Calculated Maps.

If parameters are set as described here below:

- the number of X bins (X_{bins}) = the number of the most active X block;
- the number of Y bins (Y_{bins}) = the number of the most active Y block;
- $f_1(x) = I$ (the block's i-coordinate will be displayed on the X-axis);
- $f_2(x) = J$ (the block's j-coordinate will be displayed on the Y-axis);
- Weight = the property's title (e.g., Soil – oil saturation).

The 2D histogram created will match with the calculated oil saturation property, with the z-coordinates summed up.

The number of the most active blocks along the X-axis and the Y-axis can be determined as described below:

1. in any initial property (of the type **Sum**, **Average**, or **RMS**), place the cursor on the any most remote X-axis and Y-axis block. With the cursor on the remote X-axis block, the caption in square brackets at the bottom of the image, e.g. [190, 98, 0], means that the X-axis number of the most active block is 190; the caption [13, 179, 0] means that the Y-axis number of the most active block is 179.
2. the maximum number of active blocks will match X_{max} and Y_{max} calculated automatically if **Auto Min Max** is checked.

NB. You can uncheck **Auto Min Max** and set the X_{max} and the Y_{max} one unit greater. In that case, the property will not change, but the function's value ranges will exactly match the block numbers, which will facilitate analysis and visual perception of the histogram.

11.2. X/Y Histogram

X- and Y-histograms are the X- and Y-components of the 2D histogram, respectively.

For the X-histogram (with the **Sum** type selected), the model calculates the sum of values in the vertical column of the 2D histogram's blocks corresponding to the bin, and display the sum in blue.

For the **Average** histogram, the arithmetic average will be calculated, for the **RMS** histogram, the rms of the block column values.

For an X-histogram, the blocks' range [X_{min} , X_{max}] and the X_{bins} will be saved.

For the Y-histogram, the model calculates for each bin $[Y_i, Y_{i+1}]$ the sum (or the average or the rms, depending on the histogram type) of values in the horizontal column of the 2D histogram's blocks corresponding to the bin, and display the sum in blue.

For an X-histogram, the blocks' range $[Y_{min}, Y_{max}]$ and the Y_{bins} will be saved.

Also, an X-histogram will display:

- the number of bins $[X_i, X_{i+1}]$ with at least one active block (the total amount);
- the sum of values of a parameters in all the blocks (Sum);
- the block's average value of a parameter (Average);
- the rms (of a parameter) (RMS).

X- and Y-histograms can be vertical or horizontal.

An example of a Y-histogram's application: layer distribution of mass oil-in place.

The parameters are set as follows:

- **Y-histogram;**
- **X-bins** (X_{bins}) = 1;
- **Y-bins** (Y_{bins}) = the number of layers in the models along the Z-axis;
- **X function** $f_1(x) = 1$ (only one X-bin);
- **Y function** $f_2(x) = K$ (the Y-axis will display the block's k-coordinate, ie the layer number);
- **Weight** = property title (e.g., moip – mass oil in place);
- **Type** = Sum.

The created Y-histogram shows the distribution of the current oil in place by layer of the model. You can also make the histogram horizontal; you should remember that the upper layers have smaller numbers, so they will appear in the bottom.

In the figure 169 distribution of mass oil in place among the 9 layers of the field's model is shown (9 Y-bins).

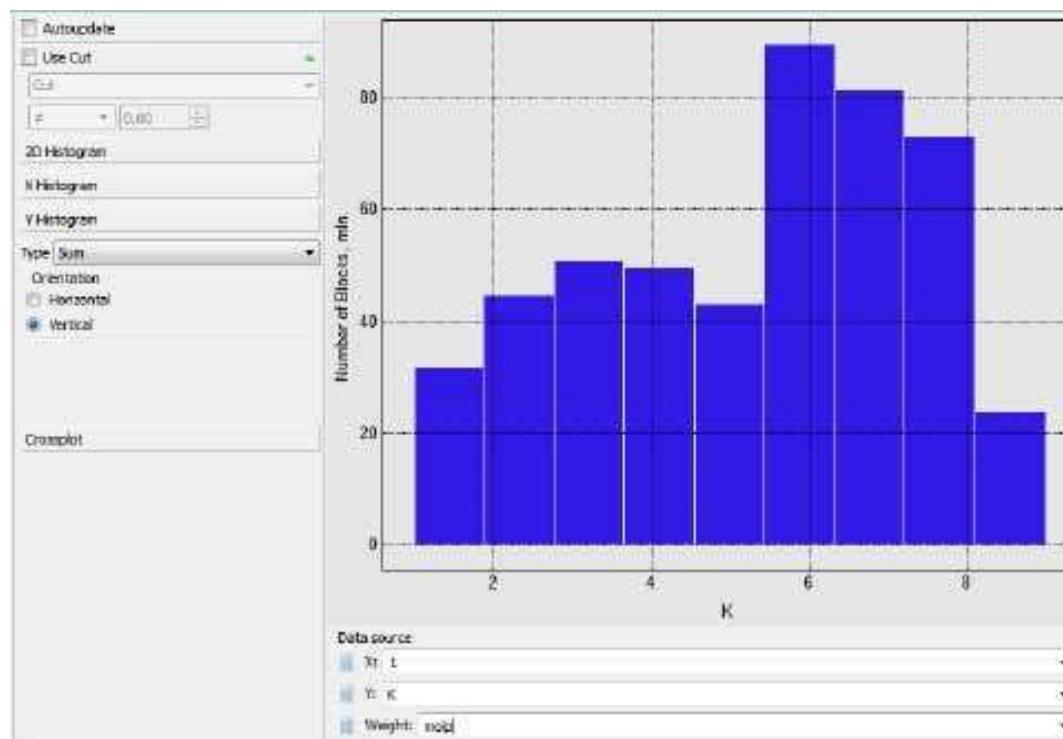


Figure 169. Y-Histogram: Distribution of moip (mass oil in place) by layers.

You can have resources displayed for certain layers only. For that purpose:

1. Create a Cut to select the layers of interest. For example: the Cut ($k==2$) | ($k==4$) will select layers 2 and 4 only.
2. In the Y-histogram, check **Use Cut**. The histogram will display the moip of layers 2 and 4 only – see figure 170

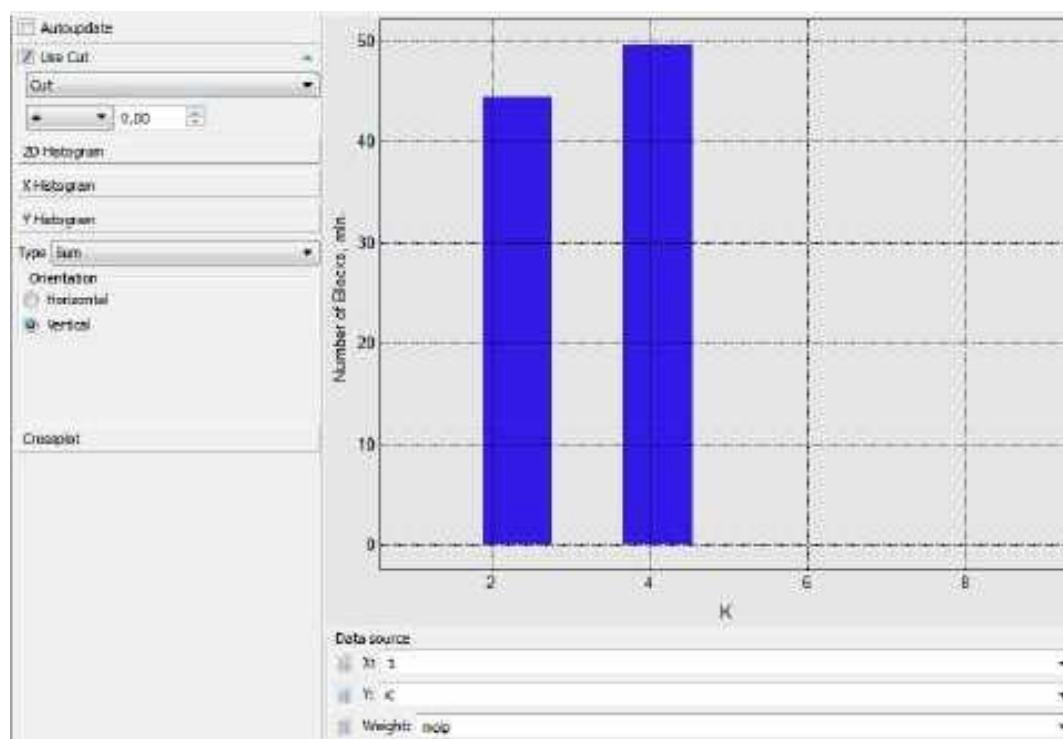


Figure 170. moip in layers 2 and 4.

11.3. Crossplot

A crossplot allows to estimate a dependence of one parameter on another. Each value of P_X parameter defined along X axis, corresponds to one or several values of P_Y parameter defined along Y axis. To create a crossplot define in the fields located at the bottom of the graph window the required parameters:

- **X** – parameter defined for X axis, e.g. porosity;
- **Y** – parameter defined for Y axis, e.g. premiability;
- **Weight** – weight function.

A crossplot is a variety of points with (P_X, P_Y) coordinates, a color of each point corresponds to a value of parameter (or a value) defined in the field **Weight**. Palette shown in the figure of crossplot (see the figure 171) sets the correspondence between colors and the values of the parameter defined in the field **Weight**.

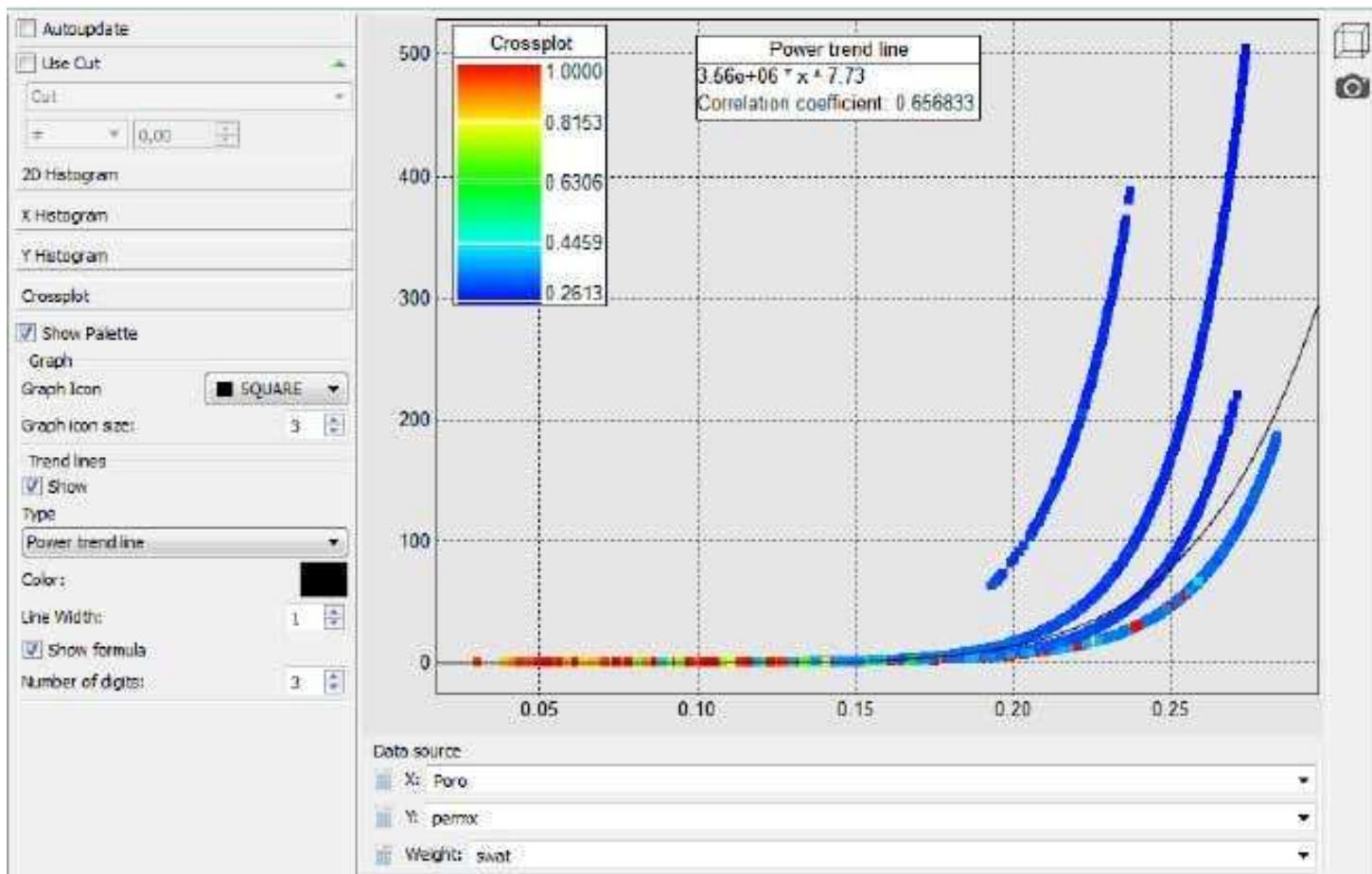


Figure 171. Crossplot for premiability vs. porosity.

The variety of points with (poro, permx) coordinates is shown in the figure 171. It can be seen that poro value equal to 0.25 corresponds to three values of permeability 50, 100 and 170. Furthermore, a color of each point shows the value of saturation of water (swat) corresponding to point's coordinates, i.e. (poro, permx) values. For example, the value of porosity equal to 0.1939 and the value of premiability equal to 62.91 correspond to the saturation of water equal to 0.26.

In order to see a trend for the created dependence of porosity on premiability check the check-box **Trend lines. Show**. The following trend lines are available:

- Logarithmic trend line;
- Exponential trend line;
- Power trend line;
- Linear trend line;
- Square polynomial trend line;
- Cubic polynomial trend line.

In order to see an analytical formula corresponding to the created trend line check the box **Show formula**. The box with the formula and a correlation coefficient will appear in the crossplot graphic. If the correlation coefficient is close to the unity then the dependence between parameters is close to the linear.

It is possible to visualize a crossplot only in the regions satisfied to the Cut filter. Follow the steps:

1. Create a Cut filter selecting the required regions, e.g. FIPNUM, region 3.
2. On the tab Crossplot check the check-box **Use Cut**. Select FIPNUM from the drop-down menu. The crossplot corresponding to the 3rd FIPNUM region will be shown, see the figure 172.

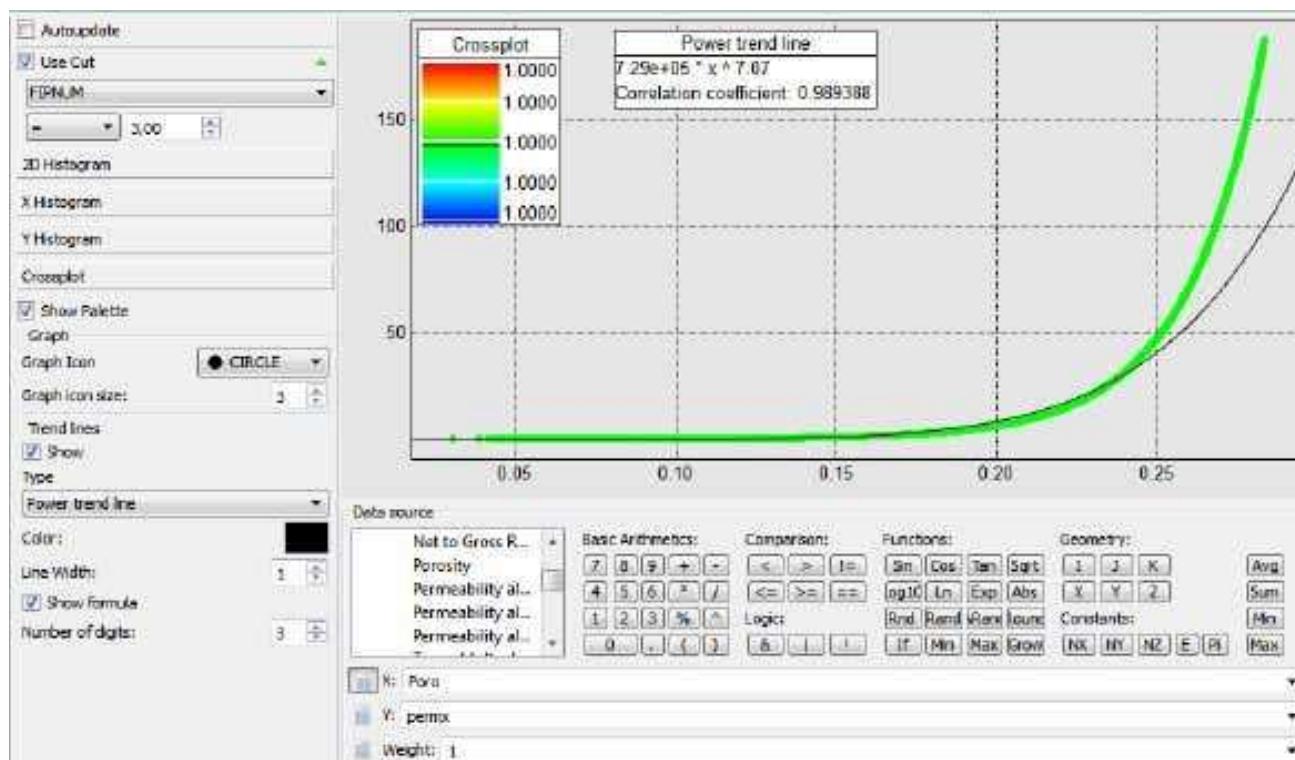


Figure 172. The crossplot for the 3rd FIPNUM region.

12. Fluid Properties

The option **Properties** shows oil-gas-water graphs and tables for different regions.

The following properties are presented (tabs in the **Properties** option):

- **RP Water-Oil:** Water-Oil relative permeability and a capillary pressure (for two-phase and three-phase models).
Note: Capillary pressure are shown relative to oil and can be either positive or negative depending on the wettable phase (hydrophilic/hydrophobic collector is).
- **COREYWO props:** Water-Oil relative permeability which is specified via Corey correlation (see the detailed description in the section Corey correlation of User Manual and keyword **COREYWO**, see 12.6.3);
- **LETWO props:** Water-Oil relative permeability which is specified via LET correlation (see the detailed description in the section LET correlation of User Manual and keyword **LETWO**, see 12.6.8);
- **RP Oil-Gas:** Oil-Gas relative permeability and capillary pressure (for three-phase models);
- **COREYGO props:** Oil-Gas relative permeability which is specified via Corey correlation (see the detailed description in the section Corey correlation of User Manual and keyword **COREYGO**, see 12.6.4);
- **LETGO props:** Oil-Gas relative permeability which is specified via LET correlation (see the detailed description in the section LET correlation of User Manual and keyword **LETGO**, see 12.6.9);
- **PVT water;**
- **PVT oil;**
- **PVT gas;**
- **PVT water with salt.** If the keyword **PVTWSALT** (see 12.7.16) is used, then two it's records **Record 1** and **Record 2** of data are visualized:
 - **Record 1** contains reference pressure (bar) and reference salt concentration in water in surface conditions (kg/m^3).
 - **record 2** contains table and graph. Salt concentration (kg/m^3), for given reference concentration value: water formation volume factor, water compressibility, water viscosity at reference depth, water viscosiblity.
- **Rock;**
- **Density;**

- SRP Oil-Water (scaled relative permeability of the water-oil system);
- SRP Gas-Oil (scaled relative permeability of the gas-oil system) (for three-phase models);
- **Component properties** (for compositional models only);
- **Components' interaction** (for compositional models only);
- **Equilibrium** (a table associated with the keyword **EQUIL**, see 12.15.2);
- Rates vs SWAT;
- Flow functions. The graph is available only if flow functions are assigned for hydrofrac and bottomhole treatment jobs in the model's data file. Also, the graph can be assigned in this tab in the GUI.
- Proppants (proppant penetration vs. pressure). The graph is available, if proppant penetration vs. pressure is assigned in the model's data file. Also, the graph can be assigned in this tab in the GUI.
- Chemical Properties;
- VFP tables for producers;
- VFP tables for injectors;
- **STANDO props** – oil PVT properties, which are set via Standing correlation (see the detailed description in the section Oil Standing's correlations of User Manual and keyword **STANDO**, see 12.5.11);
- **STANDG props** – gas PVT properties, which are set via Standing correlation (see the detailed description in the section Gas Standing's correlations of User Manual and keyword **STANDG**, see 12.5.12);
- **Molar component fracture vs. depth**, the keyword **ZMFVD** (see 12.13.17) (only for thermal models);
- **Temperature vs. depth**, the keyword **TEMPVD** (see 12.14.77) (only for thermal models);
- **Gas viscosity vs Temperature**, the keyword **GASVISCT** (see 12.14.53) (only for thermal models);
- **Oil viscosity vs Temperature**, the keyword **OILVISCT** (see 12.14.49) (only for thermal models).

Before a computation run, the model's properties can be edited – see section Properties editing.

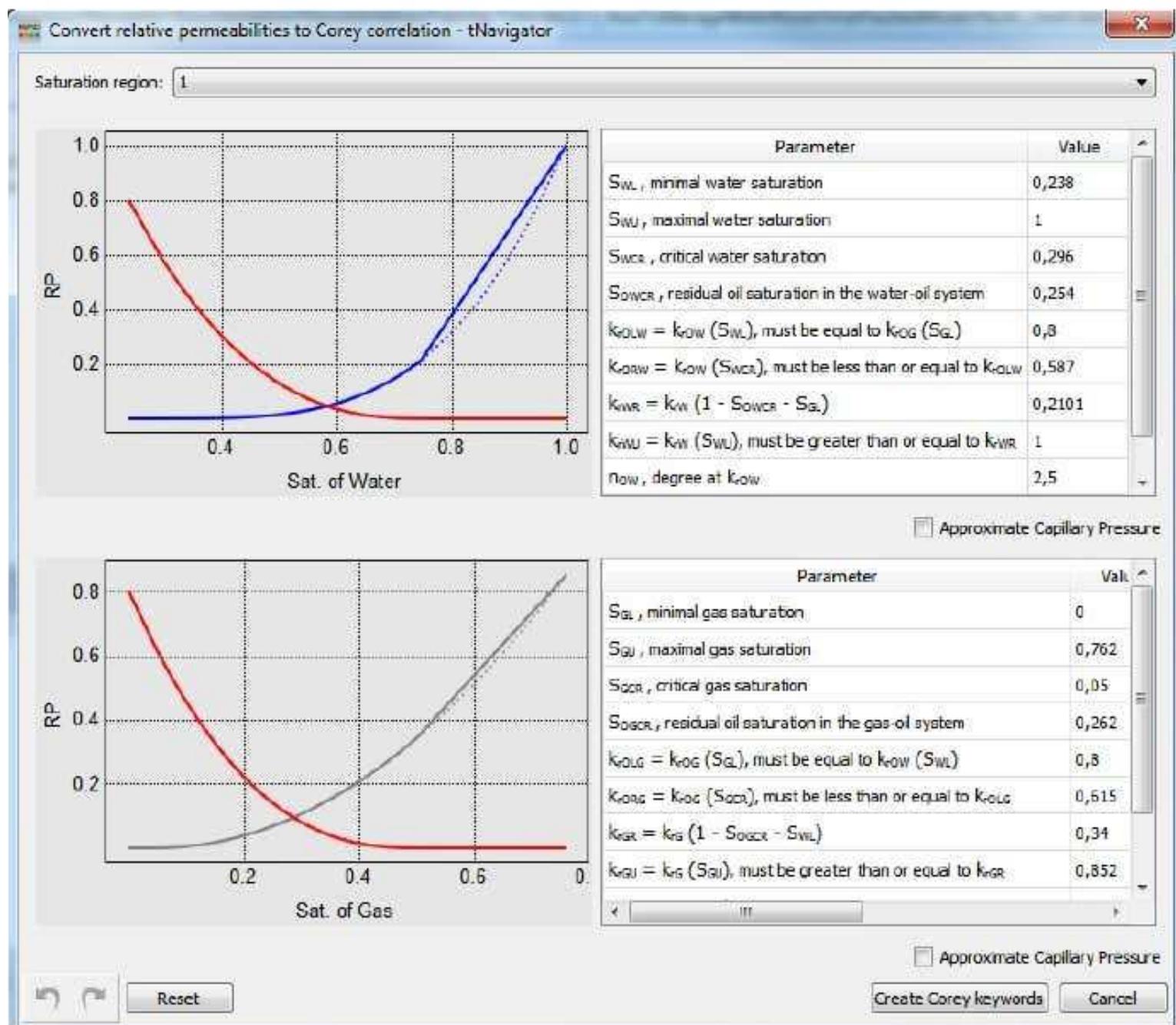


Figure 174. Conversion to Corey correlation.

12.2. Properties. Right Panel Buttons

- **Show default view.** Restores the default view of the graph. **Simultaneous click of left and right mouse buttons** do the same.
- **Export.** Exports data of pvt (pvt - properties) and rp (relative phase permeabilities) to text file (.txt). To export data, type the file name and the file path. Files <name>_rp.txt, <name>_pvt.txt with the relevant keywords (**PVTW** (see 12.5.5), **PVDG** (see 12.5.7), **ROCK** (see 12.5.17), **DENSITY** (see 12.5.25), **SWOF** (see 12.6.1), **SGOF** (see 12.6.2) and so on) will be saved.
- **Create screenshot.** See the detailed description in the section [Create Screenshot](#)

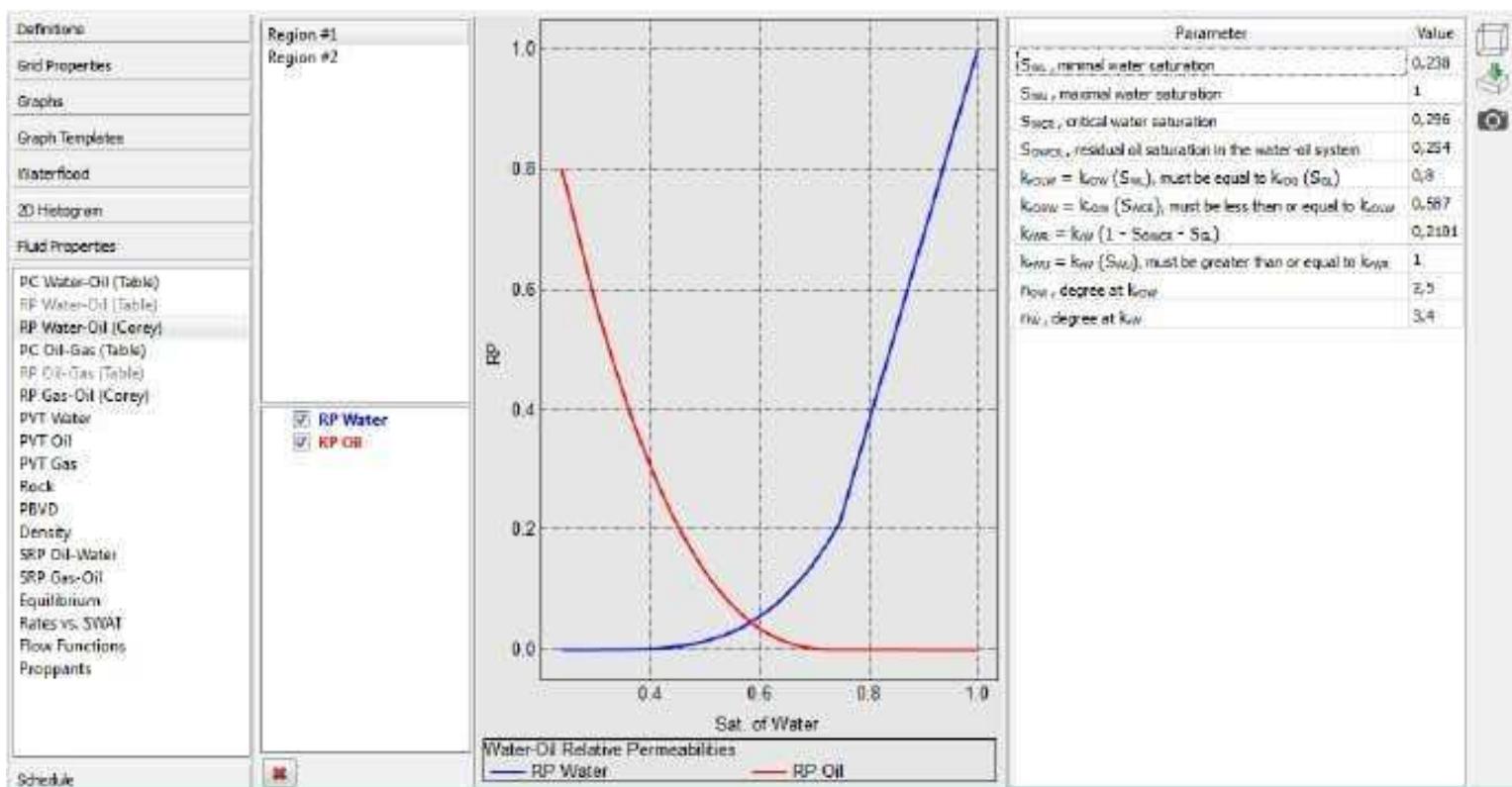


Figure 175. Corey correlation for RP in the system water-oil.

12.3. SRP (Scaled Relative Permeability Parameters)

This tab is a tool for computing and displaying permeability graphs for each block. If hysteresis option is used, then corresponding curves will be visualized – see section [Hysteresis visualization](#).

To compute permeability for block do the following:

1. In the **Block** box, assign block coordinates [I, J, K].
2. Click **Compute**.

In the figure 176 Scaled Relative Permeability Parameters for the block [3, 26, 5] are shown.

A block can be selected in 2D or 3D view. Press right mouse button on block, then select **SRP Oil-Water** or **SRP Gas-Oil**.

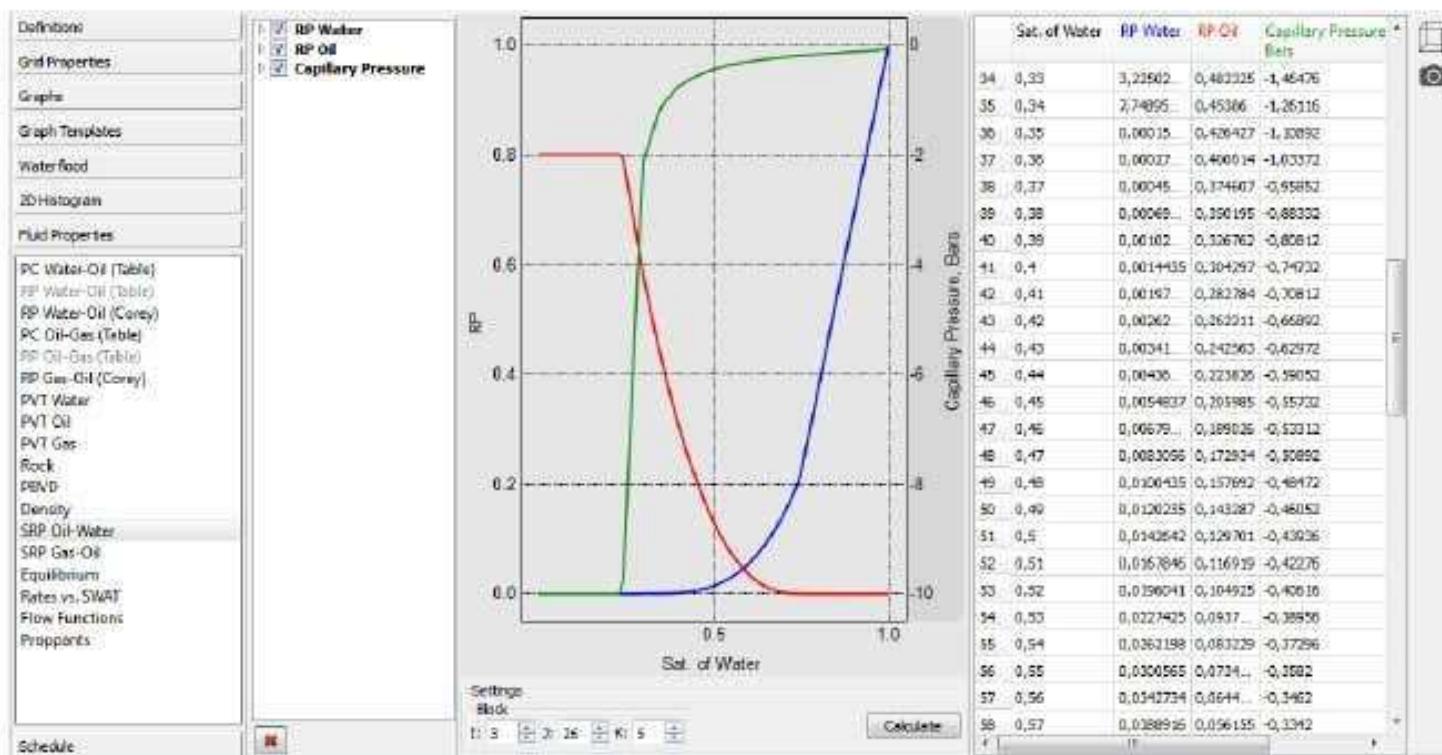


Figure 176. Scaled Relative Permeability Parameters.

12.4. Hysteresis visualization

If RP hysteresis is used in the model – see the detailed description in the section [Hysteresis](#) of tNavigator User Manual (option HYSTER of the keyword **SATOPTS**, see [12.1.71](#)), – then for RP of water, oil, gas and capillary pressure 3 curves will be visualized:

- curve of drainage process [drainage] – dotted line;
- curve of imbibition process [imbibition] – firm thin line;
- curve, which is currently used [scanning curve] – firm thick line.

Imbibition, drainage and equilibrium regions for blocks are visualized in 3D via [Regions](#).

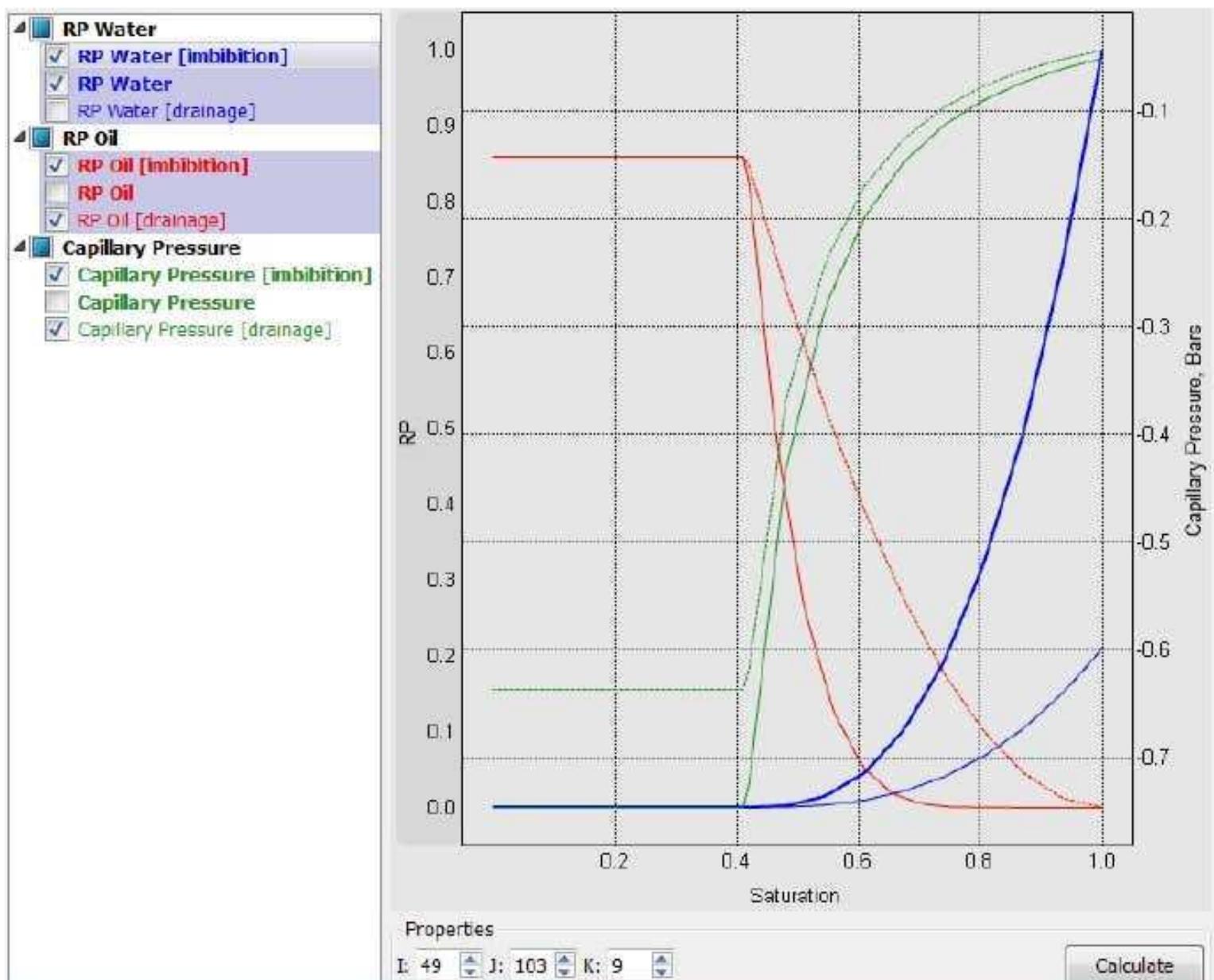


Figure 177. Hysteresis visualization.

12.5. Rates vs. SWAT

This tab is a tool for sensitivity analysis to establish at what SWAT values oil and water are produced. At each time step, the tool reviews all the blocks with all the connections and generates a histogram of oil and water volumes produced from the blocks at certain SWAT values.

To view production distribution in a computed model:

1. Place the time slider on the latest step;
2. **Properties, Rates vs SWAT;**
3. Click **Calculate**.

Graph on figure 178 shows that a large portion of the water and oil was produced at SWAT value 0.4.

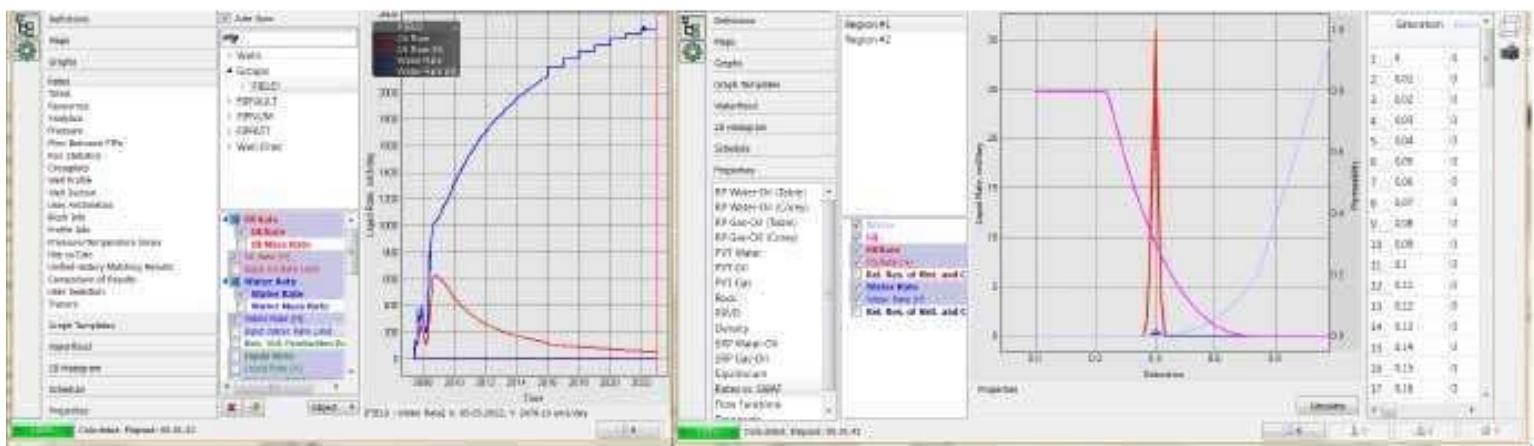


Figure 178. Rates vs. SWAT.

12.6. Flow Function

A flow function can be used to define a proppant washout and fracture clogging (during a hydrofrac job). A proppant washout and fracture sealing (a flow function) can be defined in the GUI and/or in a text file using the keywords **FLOWFUNC** (see 12.8.4) or **FLOWFTAB** (see 12.8.7) (by a list of values). The flow function's name can be specified as you assign a frac job (in the GUI or by the keywords **WFRACP** (see 12.18.129), **WFRAC** (see 12.18.127), **COMPFRAC**, see 12.18.131).

An algorithm to set a flow function in GUI is presented in the training tutorial **5.1 How To Add Fracs**. The description of model of hydrofrac is presented in the sections **Hydraulic fracture data** and **Hydraulic fractures** of tNav User Manual.

An example: a flow function (proppant washout) is defined by keywords.

For each function you should assign the name, the type, and the **k** and **a** coefficients.

- A linear function is assigned as (LIN) $F1(s) = \max\{1 + (k - 1) \cdot a \cdot s, 0\}$;
- An exponential function is assigned as (EXP) $F2(s) = k + (1 - k) \cdot e^{-as}$.

FLOWFUNC

```
'F1' LIN 0.9 0.9 /
'F2' EXP 0.1 0.1 /
/
```

Two functions have been thus assigned:

- a linear one: (LIN) $F1(s) = \max\{1 - 0,09s, 0\}$;
- an exponential one: (EXP) $F2(s) = 0,1 - 0,9e^{-0,1s}$.

The functions F1 and F2 are shown in the graph (figure 179). Option **Properties**, tab **Flow Functions**.

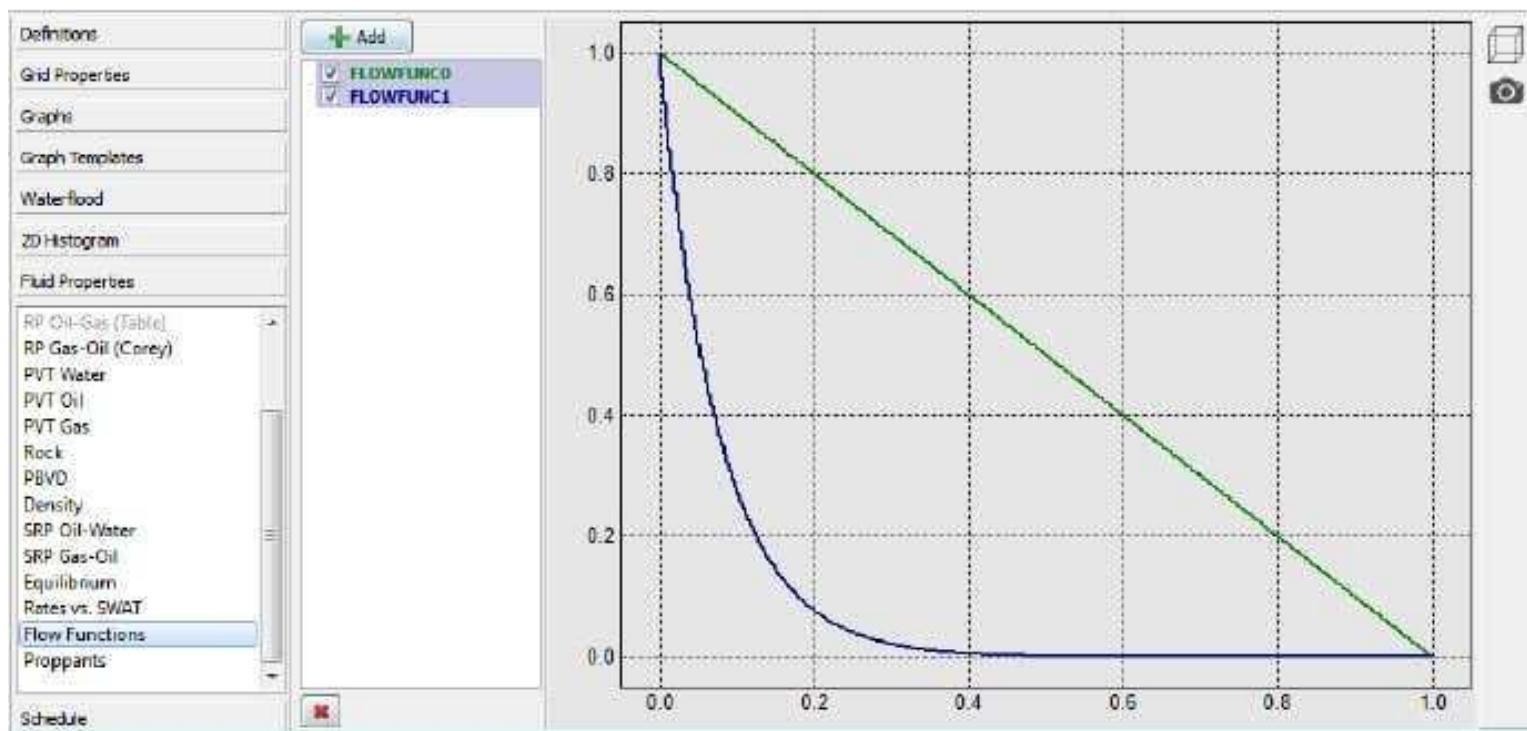


Figure 179. Flow functions Flowfunc1 and Flowfunc2 assigned by the keyword FLOWFUNC.

If flow functions are assigned by a list of values, you should specify the number of the flow functions thus assigned ([NFLOWFTB](#), see [12.8.5](#)), the names of flow functions ([FLOWFNAMEs](#), see [12.8.6](#)), and the list of values ([FLOWFTAB](#), see [12.8.7](#)).

Example: Flow functions assigned by a list of values. Two functions are assigned – F3 and F4.

NFLOWFTB
2
/

FLOWFNAMEs
'F3' 'F4' /

FLOWFTAB

0 1 1
1 0.5 *
2 * 0.5
3 0.1 0.1
/

12.7. Propants

An algorithm to set a proppant (a table of permeability vs. pressure) in GUI is presented in the training tutorial [5.1 How To Add Fracs](#).

You can assign proppant properties in the GUI (as described above) or in a text file using the keywords **NPROPANTS** (see [12.8.1](#)) (the number of proppant types in the model), **PROPANTNAMES** (see [12.8.2](#)) (proppant names), or **PROPANTTABLE** (see [12.8.3](#)) (a table of proppant penetration vs. pressure). You can assign the proppant name when you assign a hydrofrac job parameters (in the GUI or with keywords **WFRACP** (see [12.18.129](#)), **WFRAC** (see [12.18.127](#)), **COMPFRAC**, see [12.18.131](#)).

Description of the mathematical model of hydrofrac is represented in the sections [Hydraulic fracture data](#) and [Hydraulic fractures](#) of tNav User Manual.

Assigning proppant properties by keywords.

NPROPANTS

1 /

/

PROPANTNAMES

'propant1' /

PROPANTTABLE

100 100

200 80

400 20

800 10

1000 0

/

You can assign an arbitrary number of proppants. In the example below, two proppants are assigned: Proppant 12/18 and Proppant 16/20. For each proppant the penetration values at pressures from 100 to 3000 bars are specified. The Proppant Penetration vs. Pressure graph can be viewed in the option **Fluid Properties**, tab **Propants**. On the right you can see a table of pressure and proppant penetration values.

NPROPANTS

2 /

PROPANTNAMES

'proppant 12/18' 'proppant 16/20' /

PROPANTTABLE

30 1000 3000

50 900 2500

100 800 2000

150 700 1500

200 600 1300

250 500 1100

300 400 1000

350 300 900

400 200 700

800 100 100

1000 10 * /

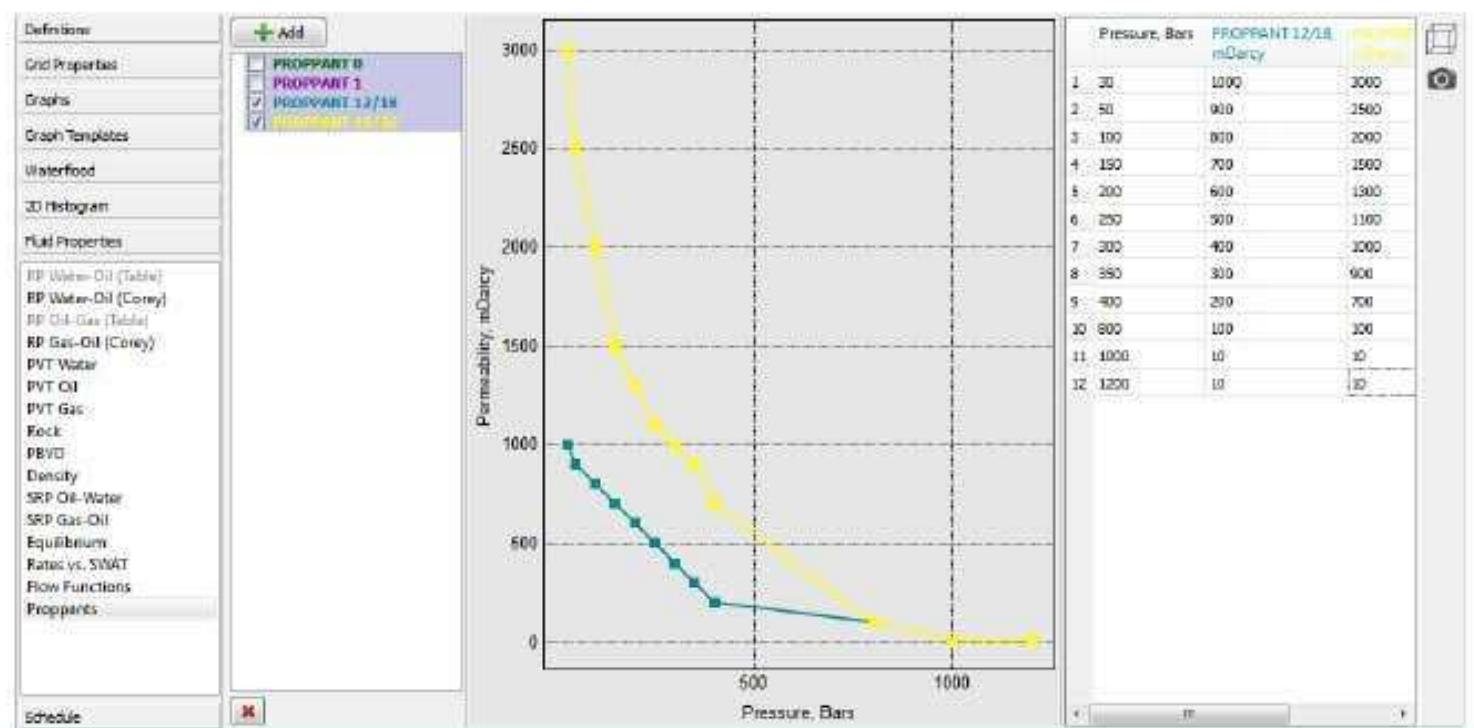
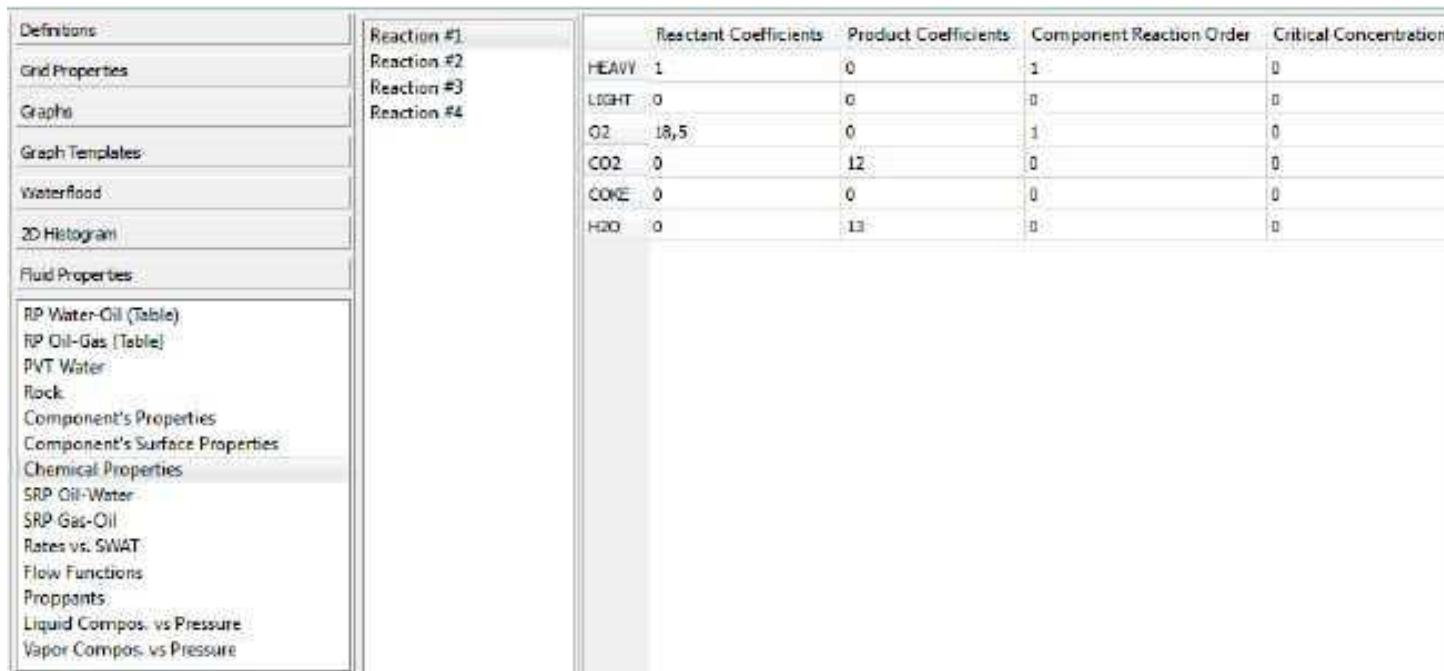


Figure 180. Graphs of proppants: proppant penetration vs. pressure.

12.8. Chemical Properties

In addition to the principal properties described in this Section, a thermal model includes **Chemical Properties** in the option **Fluid Properties**. The chemical properties are shown in the tabular form (if the model addresses chemical reactions).

For each chemical reaction (figure 181 shows 4 chemical reactions), there is a table in which the rows are model components and the columns are: reactant coefficients, product coefficients, component's reaction order, and critical concentration.



The screenshot shows the Navigator 17.3 software interface. On the left, there is a vertical navigation bar with various options like Definitions, Grid Properties, Graphs, Graph Templates, Waterflood, 2D Histogram, Fluid Properties, and several sub-options under Fluid Properties. The main area displays a table for Reaction #1. The table has four columns: Reactant Coefficients, Product Coefficients, Component Reaction Order, and Critical Concentration. The data for Reaction #1 is as follows:

	Reactant Coefficients	Product Coefficients	Component Reaction Order	Critical Concentration
HEAVY	1	0	1	0
LIGHT	0	0	0	0
O2	18,5	0	1	0
CO2	0	12	0	0
COKE	0	0	0	0
H2O	0	13	0	0

Figure 181. Chemical reactions.

If a component is not a reactant (or product), the value in the cell is **0**. In the figure 181 the reactants in the Reaction 1 are 'HEAVY' and 'O2', and the product is 'CO2' and 'H2O'.

A component interaction is set by the chemical reaction rate's dependence on reactant concentration (for example, **1** is a linear dependence).

The critical concentration of component means that if component's concentration is below the critical concentration value, the chemical reaction's rate will have a linear dependence on such concentration.

12.9. VFP tables

The description and algorithm how to set **VFP tables** for producers and injectors is presented in the description of the keywords **VFPPROD** (see 12.18.58) and **VFPINJ** (see 12.18.57) correspondingly. Also VFP tables can be calculated via correlation, which specified by the keyword **VFCPCORR** (see 12.18.62).

Select an axis to visualize, then move sliders to see a set of curves of selected parameters.

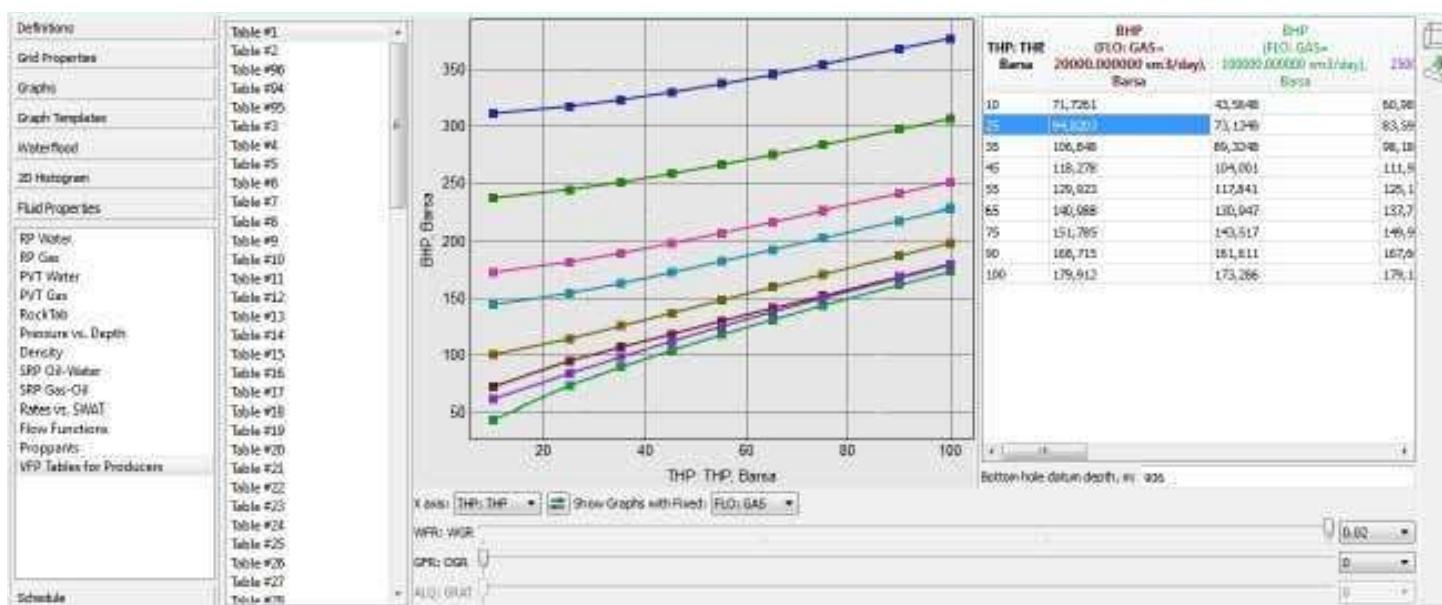


Figure 182. VFP table for producers.

13. Economical parameters

13.1. Setting Economics Parameters

Setting economics parameters:

1. In the top panel: click **Document**, select **Economics Preferences** – figure 183.
2. Select the tab **Economics**. Set economics parameters.

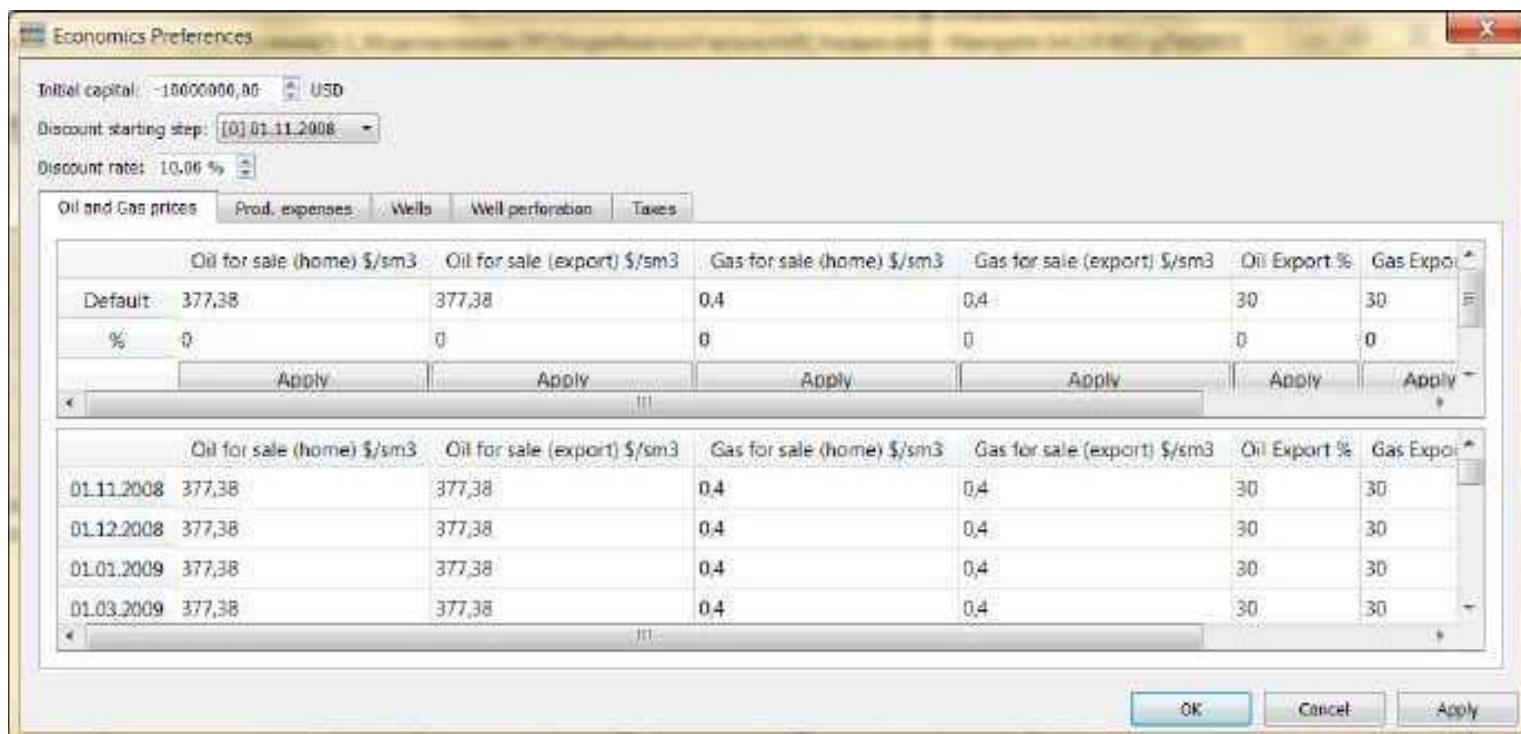


Figure 183. Preferences. Economics.

Setting Economics Parameters by keywords.

Economics parameters values can be set via keywords **ECINIT** (see 12.1.108), **ECDATES** (see 12.1.109) and **ECVAL** (see 12.1.110).

By specified values Net Present Value Graph will be built.

13.2. Net Present Value Graph

To plot a Net Present Value (NPV) graph in the course of a computation run, you should assign economics parameters (otherwise default parameters will be used).

To view an NPV graph:

1. Select the option **Graphs**.
2. Select the tab **Analytics**.
3. Check **Net Present Value**.

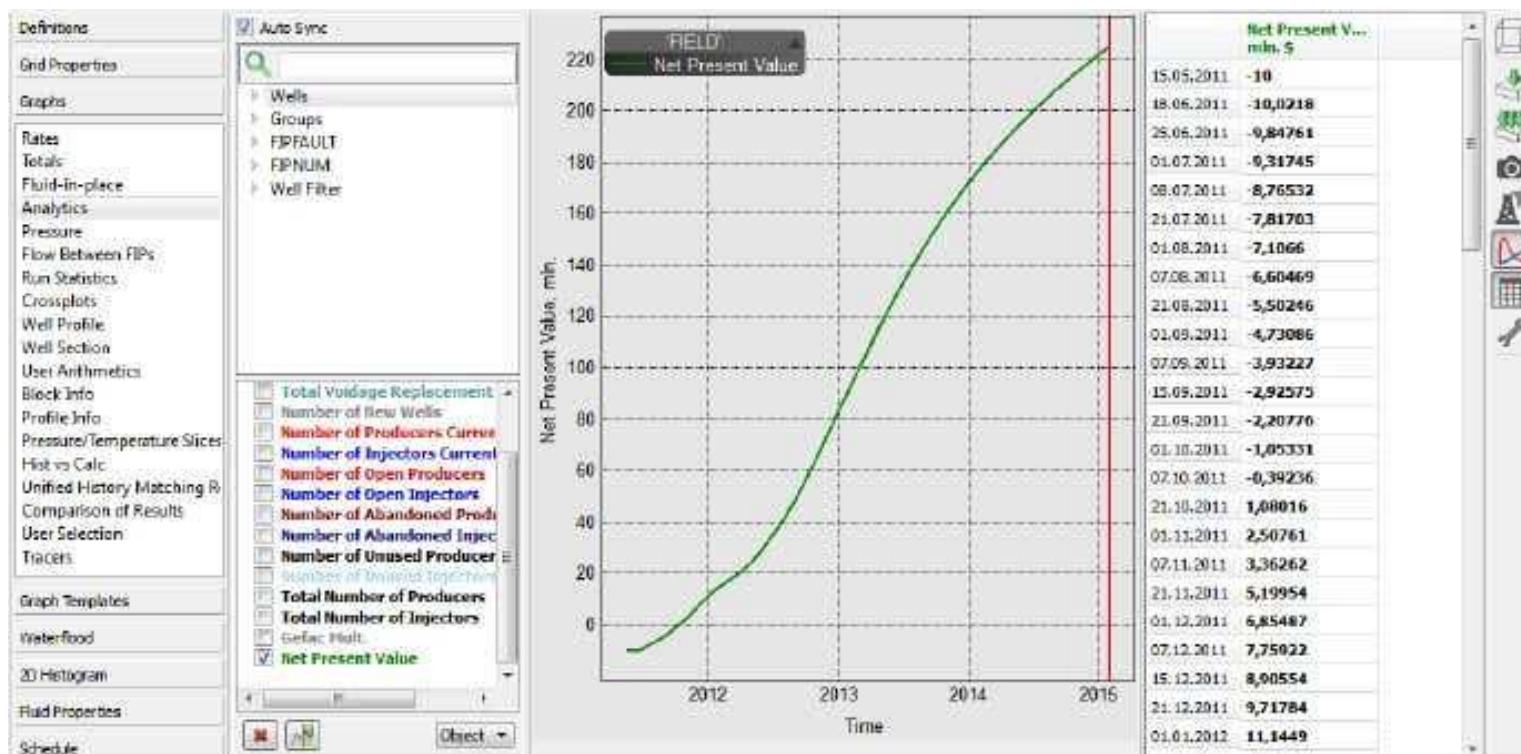


Figure 184. Net Present Value graph.

NPV Formula.

$$NPV = -IC + \sum_{t=1}^N \frac{CF_t}{(1+i)^t}$$

where:

- **CF – Cash Flow.** CF_t – cash flow in t time steps ($t = 1, \dots, N$);
- **IC – Invested Capital (Initial Capital)** at initial time moment $IC = -CF_0$ (at 0 time step)
- **i – discount rate.** It is used for the allocation of future cash flow into a single present value amount.
- **Discount starting step** – the time step in which the discount begins to be applied.

$CF_t = FI - CAPEX$, where:

- **FI - Finance income** (income from sales). Income includes income from the sale of both domestic and foreign markets (tab **Oil and Gas prices** – figure 185). FI is considered as the difference between profit before tax and profit tax.

Oil and Gas prices can increase by a given percentage each time step automatically. Specify the percent and press **Apply** – figure 185. To decrease price by a given percent you need to specify negative percentage value.

- **CAPEX – capital expenditures**. Includes the cost of drilling new wells, sidetracks. Specify cost of new well, cost of vertical, horizontal, deviated parts of the wellbore per meter – tab **Wells**, figure 186.

PBT – profit before tax: $PBT = GP - TAX - OPEX$, where:

- **GP – gross profit** (sales profit).
- **TAX – VAT (value-added tax), export duty, transport cost for export** – figure 187 (tab **Taxes**).

OPEX – operating expenditures.

OPEX = Current expenditures + Taxes and Charges.

Current expenditures include:

- Maintenance of producers (tab **Wells**, figure 186);
- Maintenance of injectors (tab **Wells**, figure 186);
- Cost of oil, gas production, water injection (tab **Prod. expenses**, figure 188).

Taxes and Charges (tab **Taxes**, figure 187):

- Salary, Insure;
- MET (the tax on mining);
- Payments for land.

14.5. Examples

14.5.1. Unary and Binary operations

Example 1. For model files:

`sgas = -swat` (unary minus, i.e. minus in front of property)

`sgas = sgas + 1` (binary plus)

Application of these two lines makes gas saturation equal to **1 – water saturation**, i.e. equivalent to `sgas = -swat` (binary minus, i.e. minus between two properties).

Example 2. For graphical interface. In **Arithmetic Command Line** the following expression can be used:

`-swat`

(unary minus, i.e. minus in front of property), the result will be the property of water saturation with `-`.

`sgas + 1`

(binary plus), the result will be the property of the gas saturation plus 1.

14.5.2. Logical operators

Examples of application of logical operators are shown below (**forms of notation when editing model data files**). In graphical interface these forms cannot be used in **Arithmetic Command Line**. In User Cuts and User Maps in the graphical interface only the right part of expressions below can be used (without `=`).

- `PERMX = 12*EXP(5*PORO)`

In this example permeability is calculated via the formula from porosity.

- `PERMX = (ARRSAT==1)*(12*EXP(5*PORO))+(ARRSAT==2)*(8*EXP(10*PORO))`

In this example permeability is calculated via the formula from porosity. In the blocks where the values of ARRSAT array is 1, the first formula is used (`12*EXP(5*PORO)`), in the blocks where the values of ARRSAT array is 2 the second formula is used (`8*EXP(10*PORO)`).

- `satnum = 1*(pvtum!=3) + 2*(pvtum==3)`

As the result of execution of this line, number of saturation region for cells from pvt regions not equal to 3 will be equal to 1, for cells from pvt region 3 saturation region is assigned 2.

- `satnum = 1*(pvtum<3) + 2*(pvtum==3) + 3*(pvtum>3)`

As the result of execution of this line, number of saturation region for cells from pvt regions 1, 2 will be equal to 1, for cells from pvt region 3 saturation region is assigned 2, and for pvt regions higher than 3 saturation regions will be equal to 3.

- $pvt_{num} = 1 * ((pressure < 200) | (soil == 0)) + 2 * ((pressure > 200) \& (soil > 0))$

As the result of execution of this line, number of pvt region will be equal to 1 in blocks where pressure is not higher than 200 or where oil saturation is zero, and 2 in the blocks where pressure is higher than 200 and oil saturation is above 0; thus, in the first region there will be all cells without oil and all with low pressure, and in the second – only cells with oil and high pressure at the same time.

14.5.3. Local changes in internal areas of a property

Graphical interface allows editing properties in whole for making user cuts or in part by means of **Arithmetics**, **Block**, **Cylinder**, **Well**, **Flow line**, **Profile** of a property editing. The detailed description of this functionality is in the tNavUserGuide section **Property Editing. Smoothing. Interpolation**.

Similar arithmetic expressions were used for setting and editing properties in the model data file. Editing for local changes in internal area of properties is done by means of the following expressions:

map(x1:x2, y1:y2, z1:z2) = expression

(where map is the edited property, expression is the acceptable arithmetic expression from constants and properties, **x1:x2**, **y1:y2**, **z1:z2** is the range, x1 is the minimum value of X, x2 is the maximum value of X, between which the property should be changed, y1:y2, z1:z2 are ranges for Y and Z axes, respectively).

If posting misses a certain interval, then the missed variable is assigned complete interval set for the model. For example, in case:

map(x1:x2, , z1:z2) = expression

Y direction will be taken completely.

Examples (forms of notation when editing model data files):

- **sgas(1:10, 2:5, 3:4) = 0**

Gas saturation with X from 1 to 10, with Y from 2 to 5 which lie in 3rd and 4th layers of the model will be assigned 0.

- **pressure(, , 4:5) = 120**

Pressure in layers 4 and 5 will be assigned 120.

- **multx(1:20,3,4:7) = 0**

Permeability factor in the cells of the specified range will be assumed as zero (vertical fault 20 cells long on X, along Y = 3, and deep from 4th to 7th layers).

14.5.4. Examples for user properties (maps)

- `soil + swat`

Visualizes a property of summation of water saturation and oil saturation.

- `dx * dy * dz * poro`

Visualizes a property of product of sizes of each cell and its porosity.

- `porv * (soil / boil + sgas / bgas * rv)`

Visualizes a property of oil reserves modified to surface conditions.

- `porv * soil / boil`

Visualizes the same property but with no account for oil in gas form.

14.5.5. Examples for user cuts

- `pressure > 300`

Visualizes cells where pressure is above 300.

- `(pvtnum == 1) & (satnum < 3)`

Visualizes cells from first PVT for which filtration area number is less than 3.

14.6. Functions for User Maps

Functions without arguments (may be used without brackets):

rand	produces scalar from 0 to 1 (this value will be assigned to all blocks)
arand	produces random values from 0 to 1 for each block

Functions of one argument:

abs	module
exp	exponent
log	natural logarithm
log10	logarithm to the base 10
sqrt	square root
sin	sine
cos	cosine
tan	tangent
min	(at the output gives scalar) minimum
max	(at the output gives scalar) maximum
sum	(at the output gives scalar) sum
avg	(at the output gives scalar) average
min_2d	(aggregating columns operator) generates at the output cylinder property: a value in each block of the vertical column is equal to the minimum of this column (it can only be used in GUI)
max_2d	(aggregating columns operator) generates at the output cylinder property: a value in each block of the vertical column is equal to the maximum of this column (it can only be used in GUI)
sum_2d	(aggregating columns operator) generates at the output cylinder property: a value in each block of the vertical column is equal to the sum of this column (it can only be used in GUI)
avg_2d	(aggregating columns operator) generates at the output cylinder property: a value in each block of the vertical column is equal to the average of this column (it can only be used in GUI)
rnd(n)	(at the output gives a property containing not more than n 1 in random blocks on the property, the rest assume zero) for MESH only
grow (property > 0,n)	displays surroundings of the set property with the radius of 1 block (for MESH only)

round	rounds a property parameter value
-------	-----------------------------------

Functions of two arguments:

min	
max	
grow (map > 0, n)	displays surroundings of the set property with the radius of n blocks (for MESH only)

Functions of three arguments:

if(condition, expr1, expr2)	returns expr1 if condition is nonzero, expr2 if condition = 0
IF-THEN-ELSEIF-ELSE-ENDIF	can be used to specify complex expression. If-then-else. In one expression ELSEIF, THEN can be used several times.

Function **box** (set the specific area of property, examples are below).

14.6.1. Examples

Using User Maps it is possible to create the property:

- **map = max (soil, swat)** (form of posting for editing a model data file) or **max (soil, swat)** (form of posting in the **Arithmetic Command Line** for User Map and in **Expression** field of Property Editing)

Visualizes a property in which maximum value between oil saturation and water saturation will be assigned to each cell.

- **round (pressure)**

Visualizes a property, where the pressure value in each cell is rounded to the nearest whole number.

- **abs (soil - swat)**

Visualizes a property of absolute value (module) of difference of oil saturation and water saturation.

- **sqrt (poro)**

Visualizes a property of square root of porosity value.

For user cuts (form of posting for editing a data file):

- **box = i < 3 & j > 5 & k == 7**

Defines the following area: Cells with numbers less than 3 on X, more than 5 on Y, and equal to 7 on Z. This posting is equivalent to the posting below **i < 3 & j > 5 & k == 7** (in arithmetic line for a User Map or User Cut **Expression** Field of property editing)

- **box = pressure > avg (pressure)**

Defines area where pressure is above the average pressure. This posting is equivalent to the posting below **pressure > avg (pressure)** (in the **Arithmetic Command Line** for a User Map or in **Expression** field of Property editing).

15.5.13. Faults

This tab is used to highlight faults assigned by the keyword **FAULTS** (see 12.2.38). The blocks in which a fault is assigned shall have the value of **1** (highlighted in red). There is a pop-down menu for you to select the fault to be highlighted or you can highlight **All Faults** – figure 216.

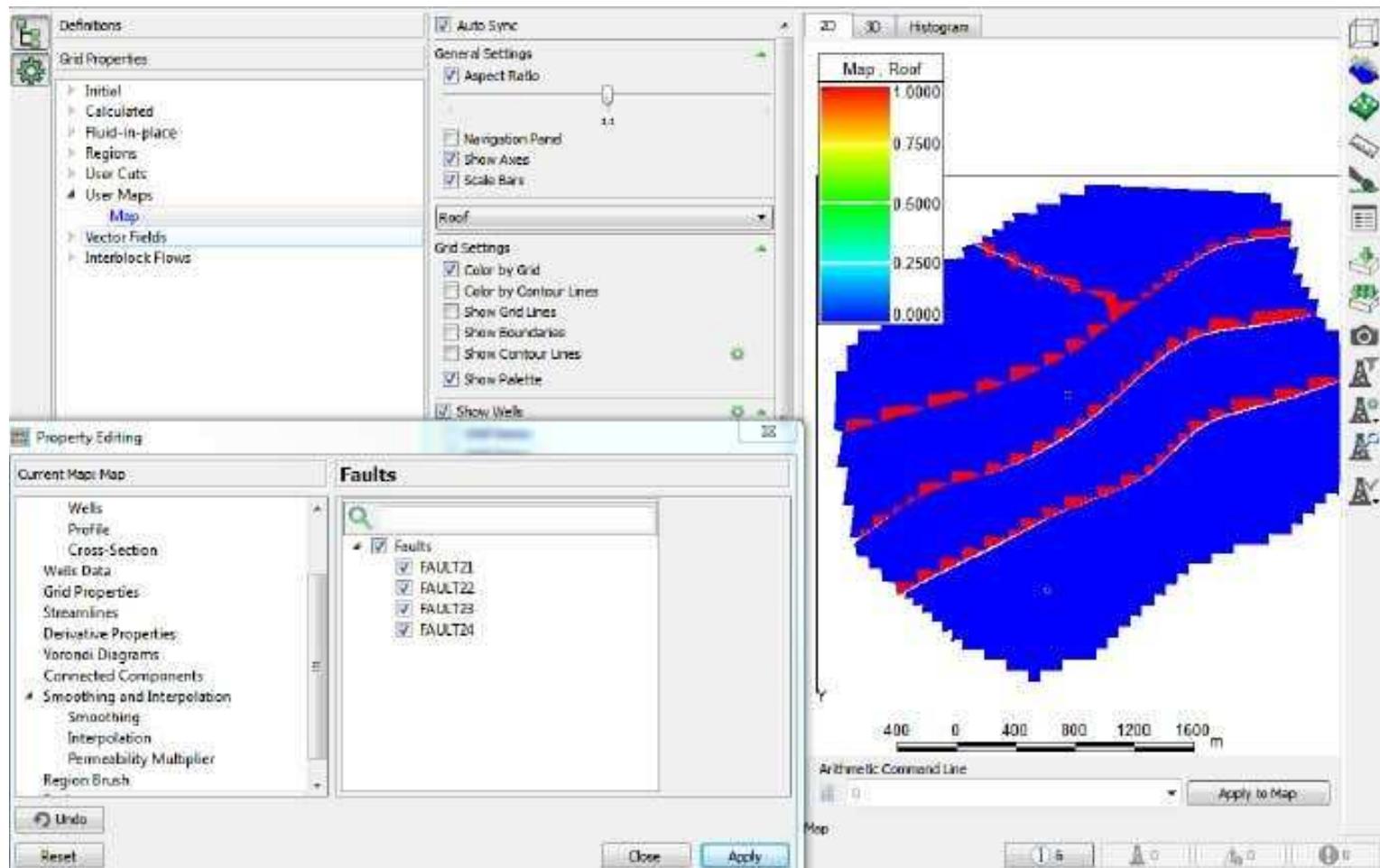


Figure 216. Map Editing. Faults.

15.6. Smoothing

Any 3D grid property can be smoothed. For a property to be smoothed, it should be opened as a user map.

See the training tutorial **4.3 How To Use Smoothing** for more detailed description.

15.7. Interpolation methods

An interpolation is a process of applying the values (e.g. porosity, permeability, uncertainty etc.) assigned to certain points to entire domain.

In tNavigator an interpolation is used:

- in graphical interface of hydrodynamic simulator to edit properties, User Maps, User Cuts via Property Editing.
- in modules Geology Designer and Model Designer to interpolate 2D Maps, horizons and grid properties.

The details of interpolation usage in hydrodynamic simulator's interface is in the section 15.8.

tNavigator supports the following interpolation methods:

- Deterministic method:
 - Least Squares method;
 - Trivial interpolation method;
 - Multilayer IDW method.
- Geostatistical method:
 - Kriging;
 - Sequential Gaussian Simulation (SGS) method.

In this section a general description of methods is given.

15.7.1. Least Squares method

There are a large number of interpolation methods. The most popular of deterministic methods is the Least-Squares method (see [3], [6]).

In tNavigator there are two possibilities of this method's implementation:

- Multilayer Least Squares method;
- 3D Least Squares method.

In the first case, the three-dimensional interpolation problem is converted to the two-dimensional one, i.e. an interpolation is carried out for each grid's layer independently.

General description of the method. Let's consider a grid, consisting of arbitrary shaped non-crossing polyhedrons (blocks) $\{b\}$ defined by 8 peaks. Some of polyhedron's peaks may coincide. For each block's peak the space coordinates (c_x, c_y, c_z) are defined. Let's N values of function F defined at arbitrary points $\{x\}$ are known: $F_i = F(x_i)$, $i = 1, \dots, N$. If a block

b_i contains a point x_i , then the value $F_i = F(b_i)$ is defined in the block. Generally speaking a distribution of points do not coincide with grid's blocks. In this case the values $F(x_i)$ are interpolated to grid's blocks. Further, for the sake of simplicity, let's suppose that values of function F are defined in grid's blocks, i.e. $F_i = F(b_i)$. In addition to a set of blocks $\{b\}$, a grid contains a set of links between blocks *links*. $linked(b_k)$ denotes a set of blocks connected with a block b_k , l_{ij} denotes a link between b_i and b_j blocks. A non oriented direction of link between blocks $Axis(l_{ij}) = (x(l_{ij}), y(l_{ij}), z(l_{ij}))$ is defined by faces, which are mutual for the blocks. An orientation of the link between blocks is defined by the function $Dir(l_{ij})$ (i.e. $x^+, x^-, y^+, y^-, z^+, z^-$). $h_x(b_i)$, $h_y(b_i)$ and $h_z(b_i)$ are the distance between mass centers of b_i block's faces along Ox , Oy and Oz , respectively.

Based on the limited set of function values the function f^* , minimizing a least mean square error of approximation calculated at the points $\{x\}$, can be defined as:

$$f^* = \sum_{i=1}^N (F_i - f(x_i))^2 + \alpha R_1(f) + \beta R_2(f),$$

where $R_1(f)$ and $R_2(f)$ are correction functions, α and β are coefficients, which defines an impact level of correction functions and varies in the range [0.01, 100]. Correction functions limit a variability of approximation values and allow to obtain smoother solutions. First and second derivatives of function f can be chosen as correction functions. $R_1(f)$ and $R_2(f)$ are computed by summation over neighboring blocks (i, j) :

$$R_1(f) = \sum_{l(b_i, b_j) \in \text{links}} w_{ij}^2 (f(b_i) - f(b_j))^2,$$

$$R_2(f) = \sum_{k=1}^N \sum_{\substack{b_i, b_j \in linked(x_k) \\ Dir(l_{ik}) = Dir(l_{ki})}} \left(w_{ik} (f(b_k) - f(b_i)) - w_{kj} (f(b_j) - f(b_k)) \right)^2 / (h_{Axis(l_{ik})}(b_k))^2$$

where w_{ij} is the weight coefficient, which can be defined differently, $l_{ij} = l(b_i, b_j)$ is a link between b_i and b_j blocks, $linked(b_k)$ are set of blocks linked with a block b_k , $h_{Axis}(b_i)$ is the distance between mass centers of b_i block's faces, quasi-orthogonal to directions $Axis = (x, y, z)$.

Depends on the chosen grid's geometry coefficients w_{ij} can be defined as:

- If $w_{ij} = 1/h_{ij}$ (where h_{ij} is the distance between mass centers of adjoining blocks), then R_1 is a sum of square of finite-difference approximations of f derivatives along directions $Axis = (x, y, z)$. R_2 is a sum of square of approximations of second derivatives .
- If $w_{ij} = 1$ the grid's geometry does not take into account.
- If $w_{ij} = T_{ij}$, where T_{ij} is transmissibility of link l_{ij} , then R_1 is computed by integration of $(\nabla f, \vec{n})$ over adjoining face of b_i and b_j blocks, where \vec{n} is the unit vector normal to the face directed to b_i block. In case of rectangular grid T_{ij} is the ratio of square of adjoining face of b_i and b_j blocks to the distance between their mass centers.

15.7.2. Trivial interpolation method

In the trivial interpolation method to each grid's block b_i , $i = 1, \dots, M$, in which a function f value is not defined, a constant value C is assigned: $f(b_i) = C$. By default $C = 0$. The assignment is carried out by layers independently.

With this method, only the cells situated along the wells will be affected by a value different from 0. If the input data are logs, an arithmetical mean (in the case of continuous property) or the most frequent value (in the case of discrete property) will be affected to the cells with several data points.

15.7.3. Multilayer IDW method

Method of Inverse Distance Weighting (IDW) is a deterministic interpolation method. IDW method is based on the idea that objects placed in the vicinity are more similar to each other than objects placed far from each other. To interpolate a value in arbitrary space point IDW method uses known values defined in the points neighbouring to this point. At the same time, the values in the points placed closer to the interpolated point have a stronger impact on the forecast value, than values in the remoted points. Thus, each point affects the forecast value only locally, and the impact decreases with increase of distance. This means that points placed close to the interpolated point have larger weights. Point's weight decreases as a function of distance. Therefore, method is called as inverse Distance Weighting method. In case of three-dimensional interpolation is carried out a dimension can be decreased to two-dimension by implementing the IDW interpolation to each two-dimensional layer of three-dimensional grid.

Let's N values of arbitrary function f are known and defined at grid's points x_i : $f_i = f(x_i)$. The interpolated value of the function f at a space point x^* is calculated by using the function's values f_i at the points x_i (interpolation nodes), $i = 1, \dots, N$:

$$f(x^*) = \begin{cases} \frac{\sum\limits_{i=1}^N \omega_i(x^*) f_i}{\sum\limits_{i=1}^N \omega_i(x^*)}, & \text{if } d(x^*, x_i) > 0 \text{ for each } i; \\ f_i, & \text{if } d(x^*, x_i) = 0 \text{ for an arbitrary } i; \end{cases}$$

where $\omega_i = \frac{1}{d(x^*, x_i)^p}$ are weights corresponding to data points, $d(x^*, x_i)$ is the distance between x^* and x_i , p is a power parameter.

15.7.4. Kriging

Kriging is a general linear regression method using statistical parameters to find optimal estimations in terms of minimum mean square deviation when constructing surfaces, properties and User Maps ([5, 4, 7]). The method is based on the principle of undisturb average value. This means that all values taken together should have a correct average value. A global undisturbness is formally provided by increase of low values and decrease of high values. To calculate unknown value of variable at a space point the Kriging method uses a variogram, a configuration of space data and values at the points in the vicinity of the selected point. A construction of variograms allows user to match a quantitative model with an available structure of space data.

In tNavigator there are two possibilities of Kriging's implementation:

- Multilayer Kriging;
- 3D Kriging.

In case of Multilayer Kriging method is used, an interpolation is carried out independently for each grid's layer, i.e. a three-dimensional interpolation problem is converted to a two-dimensional one.

The following Kriging's methods are supported:

- Simple Kriging;
- Ordinary Kriging;
- Universal Kriging.

General description of the method Let's N values of function f are known and defined at points (blocks) x_i of grid G : $f_i = f(x_i)$. A function value is assumed to be constant inside a block. A grid is a set of arbitrary shaped non-crossing polyhedrons (blocks) defined by 8 peaks. Some of polyhedron's peaks may coincide. For each block's peak the space coordinates (c_x, c_y, c_z) are defined. The aim of interpolation is to construct an interpolation function \hat{f} , which is a good approximation of unknown function f : $\hat{f}(x) \approx f(x)$ for each $x \in G$.

At a space point x^* the Kriging interpolation is linear combination of known values of the function defined at the points x :

$$\hat{f}(x^*) = \sum_{k=1}^N w_k(x^*) f(x_k)$$

Summation is carried out for known function values defined at corresponding points with coefficients w_k . w_k coefficients are calculated by solving the system of linear equations. Notice that to calculate w_k coefficients f_1, \dots, f_N values do not use. Instead, positions of points x_1, \dots, x_N and a model of probability process (variogram) are used.

It is supposed that a function f is a random function. Hence, $f_i = f(x_i)$ are random values. Then, their linear combination is a random value as well. w_k coefficients are calculated in such way that a mathematical expectation of random variable $\hat{f}(x^*)$ is equal to a

mathematical expectation of value of random function $f(x)$ at this point, and dispersion of their difference is minimal:

$$M(\hat{f}(x)) = M(f(x)), \quad D(\hat{f}(x) - f(x)) \rightarrow \min.$$

Construction of variogram.

Variogram is a key tool in a classical geostatistic, which is applied for analysis and modelling a space correlation [7]. Further the approach to construction of variogram is briefly outlined. Physical intuition suggests that values at two points, placed close to each other, are close because these values are generated under similar physical conditions (have the same "geological environment"). On the contrary, at long distance the conditions are different and greater variations are to be expected. The value variability with distance can be quantified with variogram cloud.

Let's consider known values of f at N sample points $\{x_i\}$, $i = 1, \dots, N$, for which a variogram will be constructed. All possible pairs of available points x_i, x_j , where $1 \leq i < j \leq N$, are considered. For each pair the distance $\rho = |x_i - x_j|$ and square of difference between values at these points $v = (f_i - f_j)^2$ are computed. The obtained set of points on a plane (ρ, v) is called a variogram cloud. A variogram cloud can display anisotropy (i.e. shows different behaviours along the different directions). This is frequent in 3D cases, where vertical variability is rarely of the same nature as horizontal variability (layer media). The main anisotropy directions are often suspected from geological knowledge, and a variogram cloud is calculated along these directions.

Depends on function $v(\rho)$, using to construct a curve, the following variogram models are implemented:

Exponential	$v(\rho) = c(1 - e^{-\frac{\rho}{a}})$
Spherical	$v(\rho) = \begin{cases} c\left(\frac{3\rho}{2a} - \frac{\rho^3}{2a^3}\right), & \text{if } \rho < a; \\ c, & \text{if } \rho \geq a; \end{cases}$
Gauss	$v(\rho) = c\left(1 - e^{-\frac{\rho^2}{a^2}}\right)$
Cubic	$v(\rho) = c\left(7\frac{\rho}{a} - \frac{35\rho^3}{4a^3} + \frac{7\rho^5}{2a^5} - \frac{3\rho^7}{4a^7}\right)$
Nugget-effect	$v(\rho) = c(1 - \frac{\rho}{a} \sin \frac{\rho}{a})$
Power	$v(\rho) = c\rho^a$
Cauchy	$v(\rho) = c \log \rho$
De-Vijs	$v(\rho) = c \frac{\rho^2}{\rho^2 + a^2}$

15.7.5. Sequential Gaussian Simulation (SGS) method

Sequential Gaussian Simulation method is similar to the Kriging. To get more details see [4, 7].

In tNavigator there are two possibilities of implementation of this method:

- Multilayer SGS;
- 3D SGS.

Multilayer SGS method is carried out independently for each grid's layer, i.e. a three-dimensional interpolation problem is converted to a two-dimensional one.

General description of the method. Let's define a grid G composed of arbitrary shaped non-crossing polyhedrons (blocks) $\{b\}$ defined by 8 peaks. Some of polyhedron's peaks may coincide. For each block's peak the space coordinates (c_x, c_y, c_z) are defined.

Let's consider known values of function f at N sample points x_i of grid G : $f_i = f(x_i)$, $i = 1, \dots, N$. A function value f_i is assumed to be constant inside a block.

A process of variogram construction for this method coincide with construction in method Kriging.

In contrast to Kriging method, for the SGS method the result of interpolation at point x^* is a linear combination of defined number of points N_k (where N_k is **the number of kriging points**) selected in the region limited by **Kriging Radius**.

A summation is carried using known values of function f defined at points x_i :

$$\widehat{f}(x^*) = \sum_{i=1}^{N_k} w_i(x^*) f(x_i)$$

w_i coefficients are calculated by solving a system of linear equations.

15.8. Interpolation

Interpolation is a process of applying the values assigned in certain points (or certain blocks of the model) to the entire grid (to all blocks of the model).

tNavigator supports the following interpolation methods, which can be selected from a pop-down menu – see figure 217:

- Multilayer Least Squares;
- 3D Least Squares;
- Multilayer Kriging;
- 3D Kriging;
- Multilayer SGS;
- 3D SGS;
- Trivial;
- Multilayer IDW.

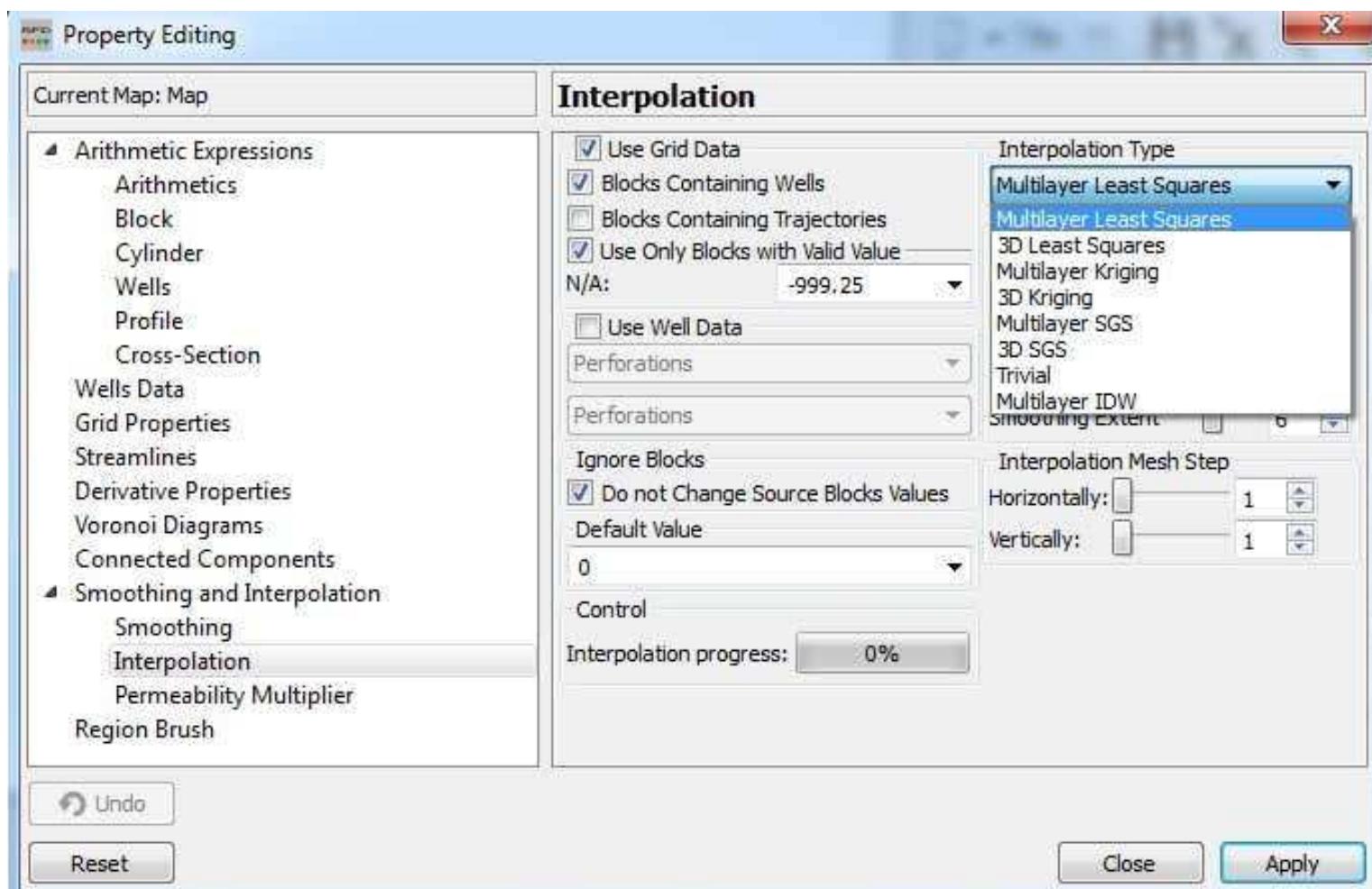


Figure 217. Property Editing. Interpolation.

You can set interpolation preferences via GUI and model data files using the keyword **INTERPOLATE** (see 12.3.13).

Examples of implementation of interpolations are given in training courses **4.2 How To Use Arithmetic** (via keyword) and **4.4 How To Use Interpolation** (via GUI).

Available Interpolation options are the following: **Use Grid Data** or **Use Well Data** (check an option you need).

Use Grid Data.

- Blocks containing wells (interpolation nodes are blocks that contain well connections (perforation intervals) with a property Map value in the block)).
- Blocks containing wells' trajectories (interpolation nodes are blocks that contain wells' trajectories with a Map value in the block)).

Use Well Data.

The values of the selected parameter for each well connection (perforation interval) are projected to the block with the well connection (perforation interval). Then the values in the blocks with connections (perforation intervals) are interpolated to the whole **Map**.

For example, if you select **Mismatches, Oil Total (Mismatch)**: the historical computed value of cumulative oil for each well connection (perforation interval) is projected to the block with the connection (the perforation interval). Then the values of the blocks with connections (perforated intervals) are interpolated to the whole **Map**.

The following parameters are available to be selected:

- Well logs;
- Rates;
- Cumulative production (Totals);
- User Maps;
- Initial;
- Calculated;
- Analysis;
- Pressure;
- Connections;
- Mismatches.

You can select a **default** value – the value assigned to the layer, if the layer contains no interpolation nodes. The initial default value is **0**.

15.8.8. Interpolation by multilayer IDW method

In this section a specification on implementation of the interpolation method to edit a property **Map** is given. A general description of the method, formulas and details of the use of multipliers and coefficients, mentioned in this section, are given in the section Multilayer IDW method.

In order to implement the Multilayer IDW method follow the steps:

1. Right-click by **Map. Edit**, select **Interpolation**;
2. **Multilayer IDW**. Set parameters and power parameter.

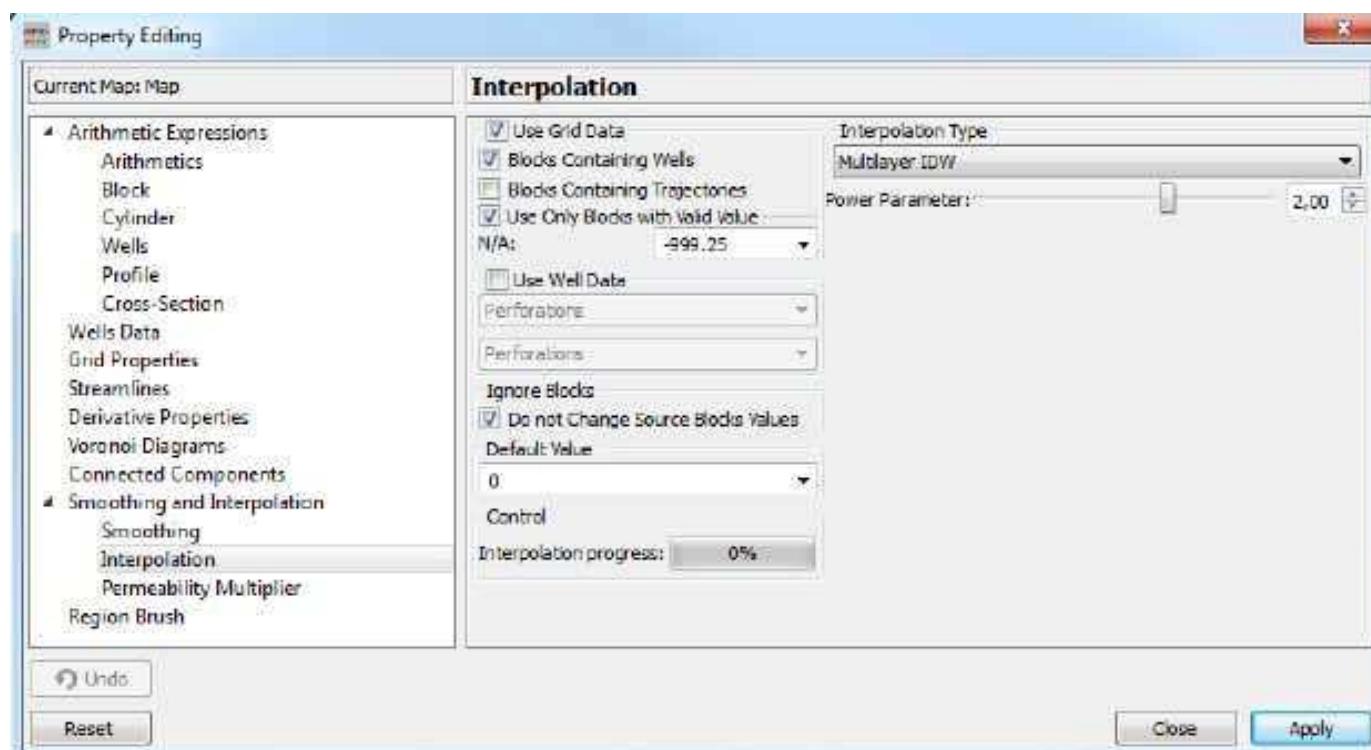


Figure 227. Multilayer IDW.

15.9. Permeability Multiplier

This tab in the Property Editing can be used to convert a **User Map** (an interpolated property, for instance) into a **MultX** property (an X transmissibility multiplier).

A new property **NewMap** is created as follows:

$$\text{NewMap} = e^{\alpha \text{Map}}$$

where α is the multiplier (to be assigned by the **Degree Coefficient** slider).

Bounds of Multiplier Value: the minimum and the maximum permissible value (if the computed value exceeds the maximum, the maximum value assigned is used). In the case, which shown on figure 228, 0 is replaced by 1, the minimum value is 0, the maximum value is 15 (so the MULTX will be 0 to 15).

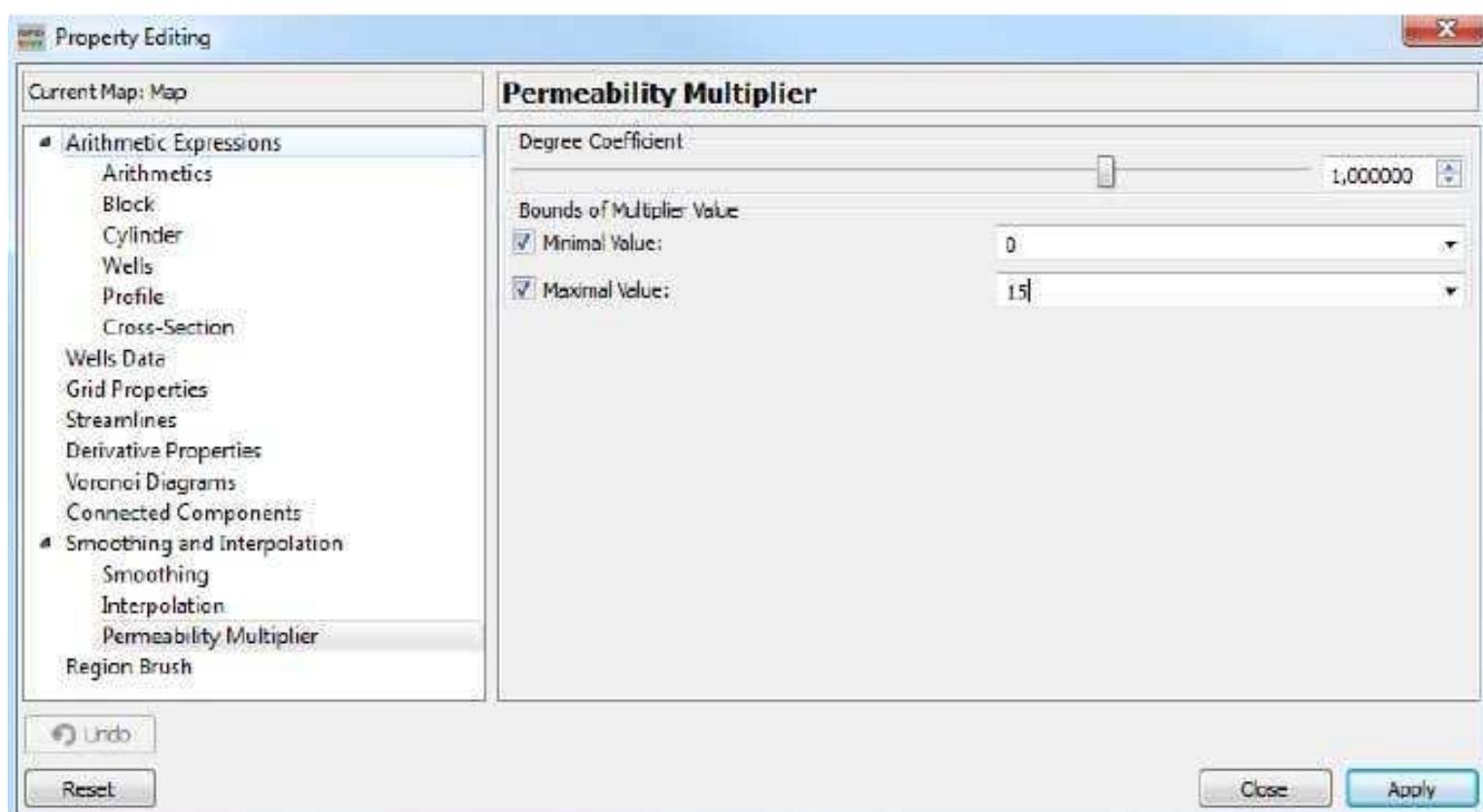


Figure 228. Property Editing: Transmissibility Multiplier.

16. Field Development Planning

In GUI you can do the following operations for Field Development Planning:

- Add a vertical well;
- Add a horizontal or deviated well;
- Add a side track;
- Edit Well Properties;
- Export well trajectories in WELLTRACK format (X, Y, Z, measured depth);
- Set tracer injection;
- Create a Forecast Model;
- Create hydraulic fracture;
- Set well bottomhole zone treatment;
- Load wells data (trajectories, groups, events, history, RFT (MDT), PLT). Data can be loaded via the menu **Document. Load Well Data**. Formats are described in the section 9.

16.1. Adding a well. Forecast. Tracers

You can add wells prior to a computation or at any step during a pause in computation. Wells are visualized as they drilled. Even if the model has not been computed yet, you can display all the wells if you move the time slider to the last time step or checking **Show All Wells**. Wells not yet in operation at the current time step will be shown in gray.

The default command for adding a well / group of wells is **Alt+Click**. To edit the properties of an existing well, put the cursor on the well on a 2D Map or 3D Map and press **Ctrl+Click**.

Detailed description of the following features is presented in training course **1.2 How To Do Field Development Planning**:

- Add a vertical well;
- Add a horizontal or deviated well;
- Add a well trunk;
- Export well trajectories in WELLTRACK format (X, Y, Z, measured depth);
- Add a well as per a well pattern;
- Add new wells with open connections in certain layers only (with high oil saturation or with number of Z-layer in specified range).

The detailed description of creating of forecast model in GUI is presented in the training course **1.6 How To Use Restart**.

Basic steps to create a forecast:

1. Set the time slider on the previously computed time step from which the forecast is to start.
2. On the top panel, click **Document**, **Create Forecast**.
3. In the dialog **Create Forecast Model**, set the required parameters.

The detailed description of setting tracer injection and tracer graphs in GUI is presented in the training course **2.2 How To Interactive Tracer Injection**.

16.2. Hydraulic fracture

A frac job is described by the following parameters:

- The well's name and the frac job's date;
- The properties of the proppant used (penetration vs. reservoir pressure) (the number of proppant types in the model is assigned by the keyword **NPROPANTS** (see 12.8.1), the proppants' names by the keyword **PROPANTNAMES** (see 12.8.2), and the table of proppant properties vs. pressure – by the keyword **PROPANTTABLE**, see 12.8.3). This is assigned in the option **Properties. Proppant**;
- Proppant washout is a function of the fracture penetration vs. phase flow or time (assigned by the keywords **FLOWFUNC** (see 12.8.4), **FLOWFTAB** (see 12.8.7), **FLOWFNAMES**, see 12.8.6). This is assigned in **Fluid Properties. Flow Functions**;
- induced fractures' azimuth – φ ;
- induced fractures' half-length – L ;
- fracture aperture (fracture width at the well) – w ;
- height (the numbers of the first block and the last block penetrated by the well path) – h ;
- fractures' slow angle – ψ .

Full description of the mathematical model of Frac Job is presented in the section Modified well model of tNavUserManual.

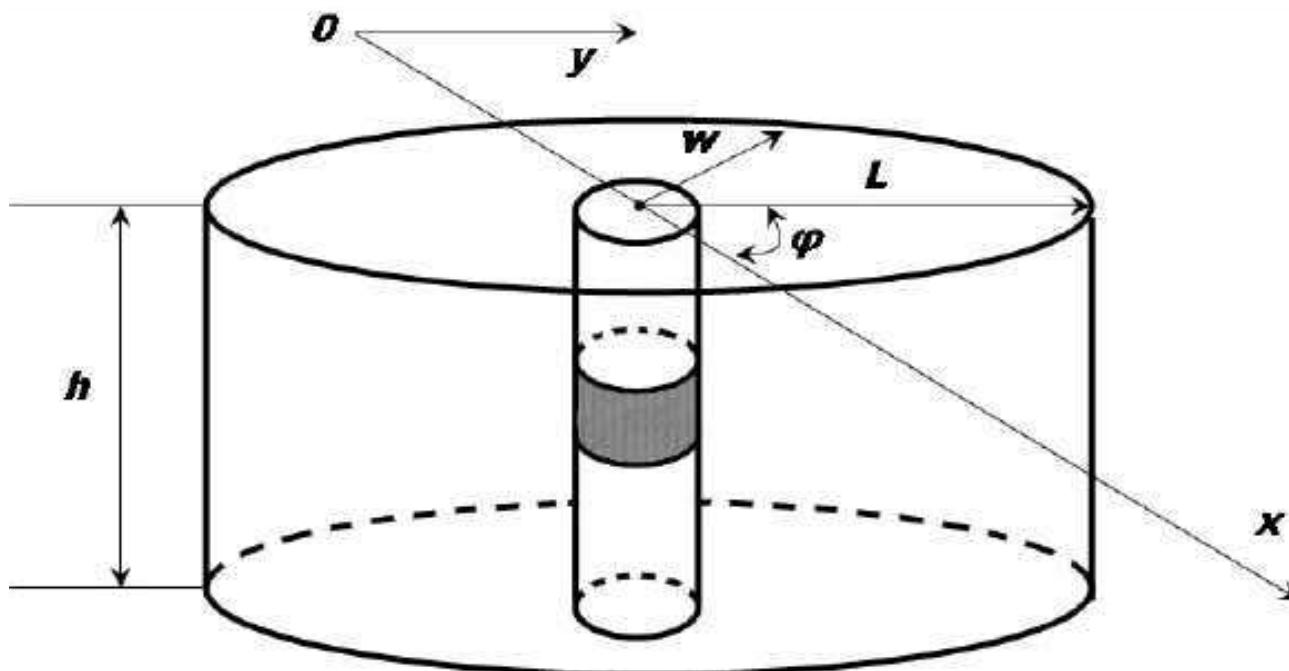


Figure 229. A scheme of Frac model.

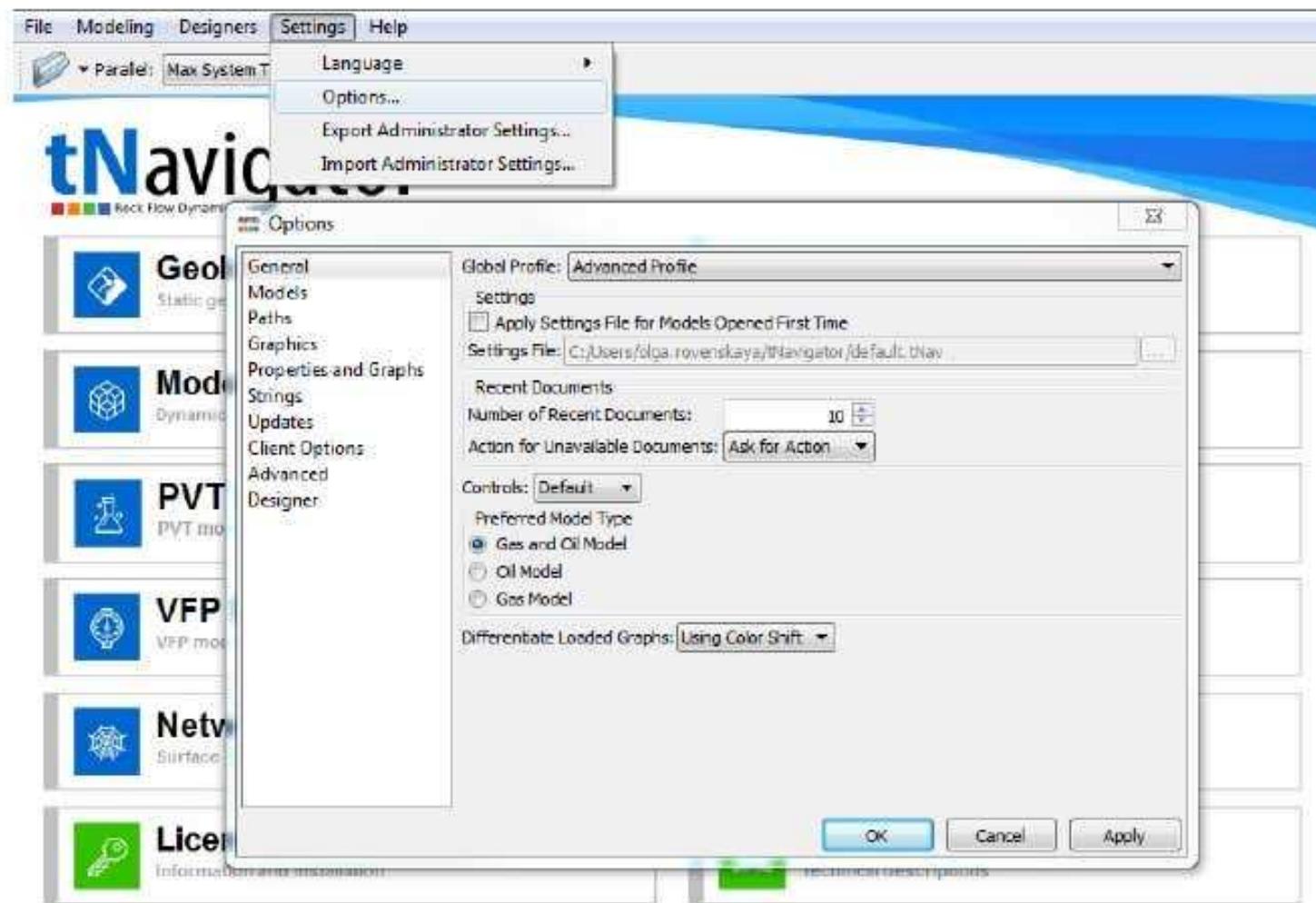


Figure 231. tNavigator's Options.

17.1. tNavigator's General Settings

To open tNavigator's basic options, go to the main menu, click **Settings** and select **Options** in the pop-down menu. Go to the tab **General**.

The tab's settings are:

1. Global Profile

- Advanced Profile (with all the existing options, all properties and graphs).
- Simple Profile (some options, properties, and graphs have been removed to simplify the use of tNavigator).

2. Settings File: Apply settings file for Model Opened First Time.

3. Recent Documents (recent documents (projects) handling parameters):

- The number of recent documents available (the number of documents that can be opened from the list in the option **File, Recent Document**). The default number is 10.
- Actions for unavailable documents (the pop-down menu options: Exclude Record, Ask for Action, Do Nothing). The default action is **Ask for Action**. If the latest

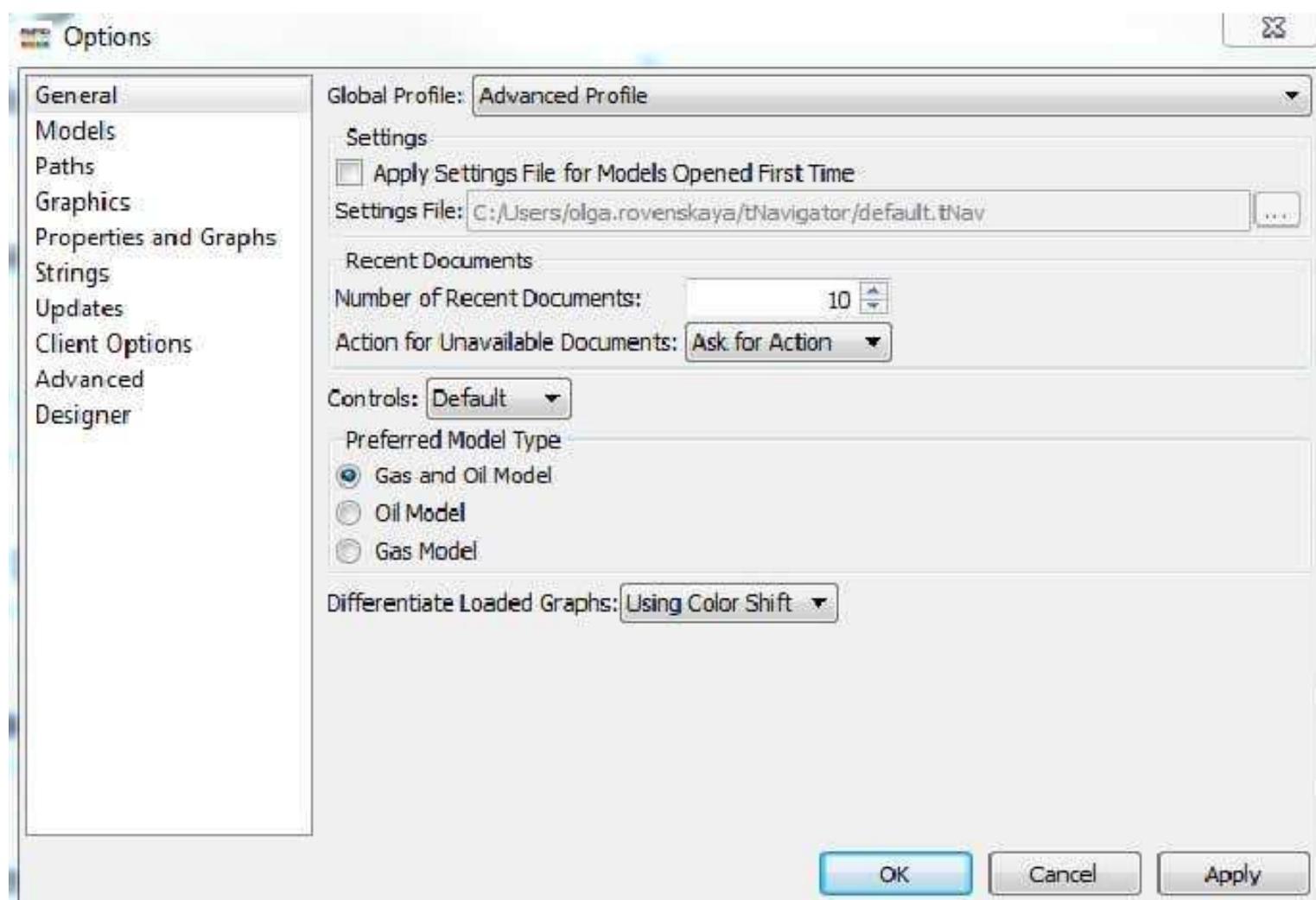


Figure 232. tNavigator's General Settings.

recent document has been deleted or moved to a different location, you will see window below (see figure 233) when trying to open that document from **File, Recent Documents**:

Actions available: **Delete record from Recent Documents list, Indicate new location of the document, or Save the record.**

4. **Controls:** Default, Petrel, IRAP RMS (select in the pop-down menu). This helps make settings for scaling and movement of 2D and 3D visualizations in accordance with the programs' control buttons (the default controls are the tNavigator buttons).
5. **Preferred Model Type:** Gas and Oil Model, Oil model, Gas model (this type defines which properties and graphs will be checked for visualization by default).
6. **Differentiate Loaded Graphs:** using icons, using color shift. Sets the difference in visualization between the graphs. The detailed description how to load graphs is given in the section [Multiple Models' Results Graphs in the Same Window](#)

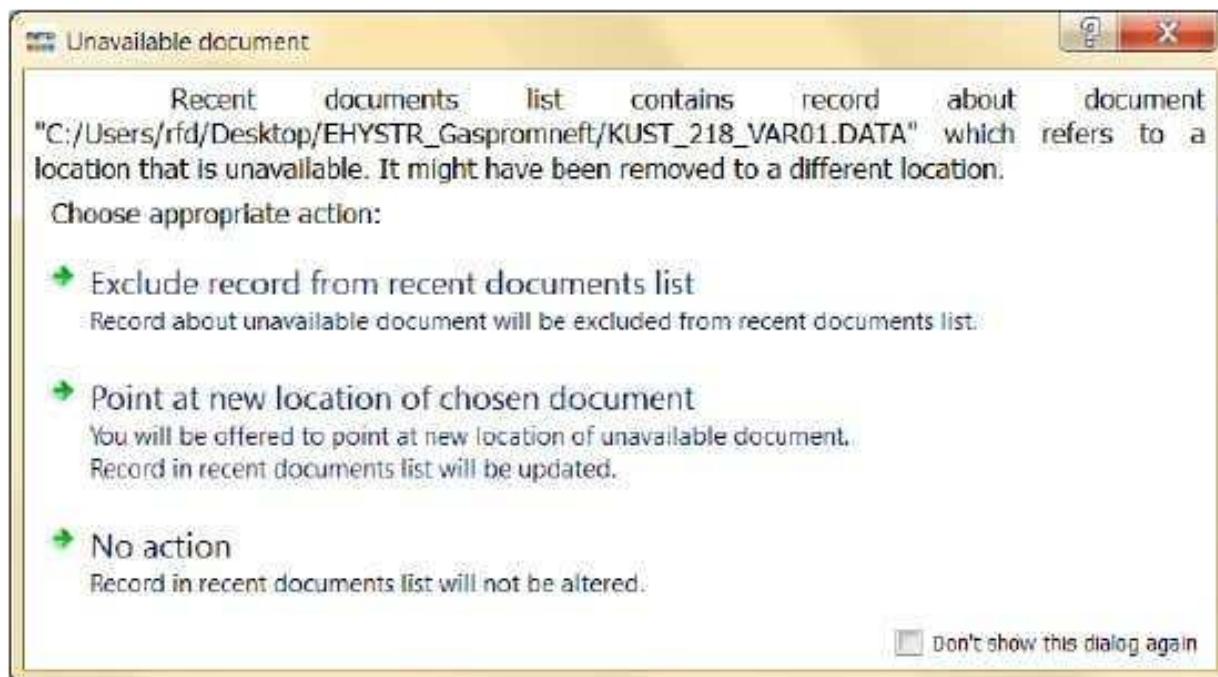


Figure 233. Select the action when trying to open an unavailable document.

17.2. Models

Saving/Loading Models (Load/Save parameters).

- **Check and Load MORE Results Files.**

If Eclipse or Tempest MORE results files have been loaded, you will have Eclipse or Tempest MORE results graphs with the above names marked [E] for Eclipse or [M] for Tempest MORE). For example, **Oil Rate [E]** means oil rate computed by Eclipse.

- **Check and Load Eclipse Restart Files.**

- **Automatically Load User Files** (auto reads user files from the USER subfolder on Model Load).

If this option is not checked, then you will be asked while opening model to specify which files in the USER subfolder should be loaded and which should be ignored.

See the detailed description in the section [USER folder](#) of tNavUserManual.

- **Automatically Save User Files** (auto saves user files to the USER subfolder). If this option is not checked, you will be asked when closing the model whether the files with new events and properties should be saved to the USER subfolder. User files autosave procedure is described here.

- **Automatically Run Model on Open** (check this option if the model does not have to be edited or viewed prior to a computation).

- **Save Intermediate Eclipse Model** (non-Eclipse models only). This will save interim model files for IMEX, STARS, and MORE models in Eclipse syntax.

- **Don't Hide Multiple Messages on Conversion.**

- **Show Selective Writing of Results Wizard on First Open.**

Setting for a selective recording of steps if necessary (this can be selected when opening the model – Results Writing Wizard). Selective recording of results to the RESULTS sub-folder. You can have all the data for all steps recorded (the default setting) or have only some data for some time steps recorded.

Describing of Selective Writing of Results Wizard is represented in the section [1](#).

- **Use Compressed Format for User Maps Saving on Model Close.**

- **Write Initial Properties.**

To save and split the model it is necessary to check this setting. Recording of initial properties is not done by default to speed-up models' opening on slow shared disks and to reduce the size of the folder with calculation results.

- **Show limits in Historical Graphs.**

If this option is activated specified limits will be visualized in graphics.

- **Keep inactive wells in MORE models.**

If this option is activated wells located outside a reservoir are not taken into account when calculating parameters based on historical data and visualizing number of wells.

- **Default Input Syntax for Data Files** (pop-down menu options: E100, E300). By default, all the Data-models will be opened in the format selected.

- **Default Input Syntax for Dat Files** (pop-down menu options: IMEX, GEM, STARS, MORE). By default, all the Dat-models will be opened in the format selected.

17.3. Paths

- Editor.** You can assign a text editor by entering the full path to its executable file (e.g., **C:/WINDOWS/Program Files/Notepad++/notepad.exe**). In this case files from **Files** menu will be open via this editor.

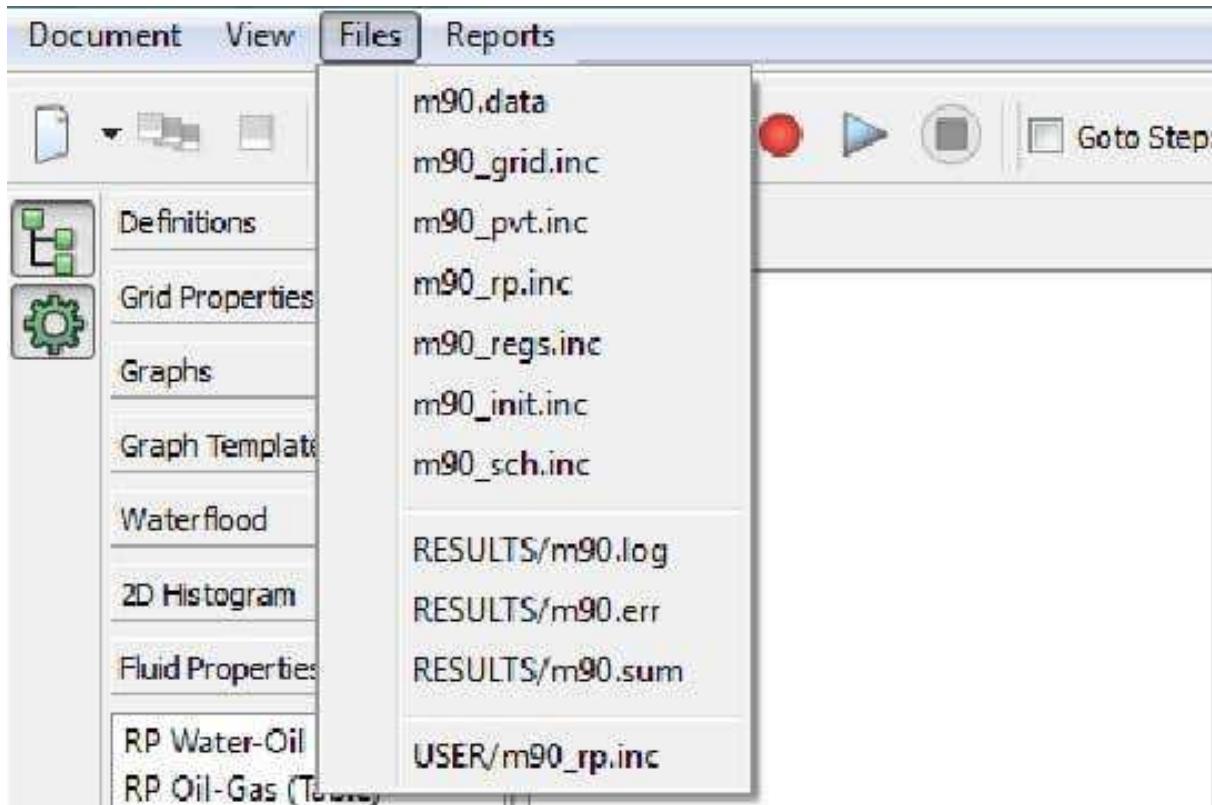


Figure 234. Open model files in tNavigator.

- Console version.** Specify path to exe-file of console version.

Detailed description of console version is given in the section [tNavigator Console version](#) of tNav User Manual.

- PDF-viewer** (select pdf-viewer to view manuals files opened using menu **Manuals** of tNavigator main window or option **Help** of top menu of main window). Specify full path to exe-file of non-default PDF-viewer.

17.4. Graphics

To open tNavigator's graphics and fonts options, go the main menu, click **Settings** and select **Options** in the pop-down menu. Go to the tab **Graphics**.

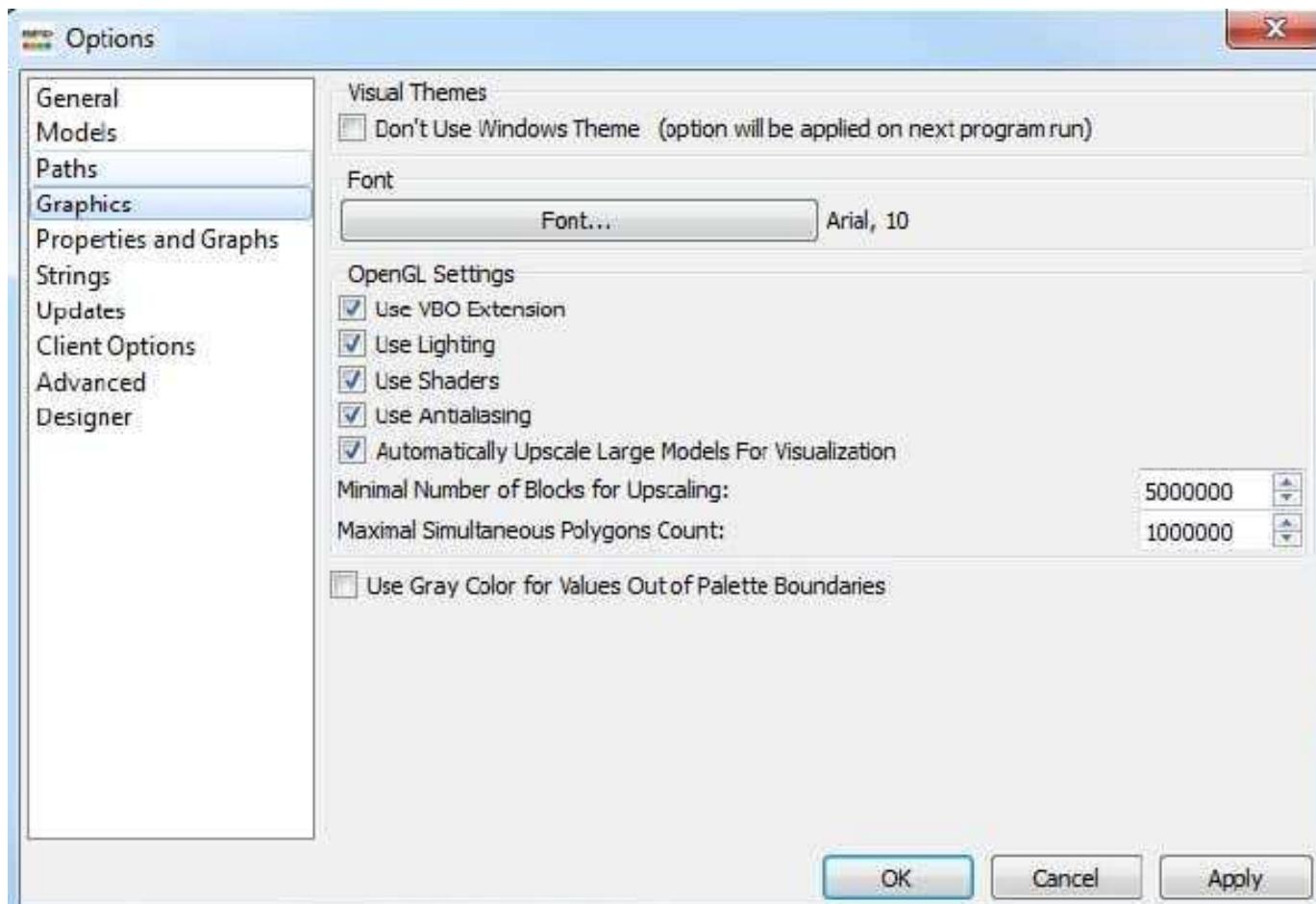


Figure 235. Graphics and Fonts dialog.

The tab's settings:

1. Visual Themes:

- Don't Use Windows Theme (option will be applied on next program run)

2. Fonts:

- Font, font style, size, effects, writing system;

3. OpenGL Settings:

- Use VBO;
- Use Lighting;
- Use shaders.
- Use Antialiasing;
- Automatically Upscale Large Models For Visualization;
- Minimal Number of Blocks for Upscaling;

- Maximal Simultaneous Polygons Count.
- 4. Use Gray Color for values Out of Palette Boundaries.**

17.5. Maps and Graphs

To open the dialog for settings for properties and graphs to be displayed and properties edited in the GUI, go to tNavigator's main window, click **Settings** and select **Options** in the pop-down menu. Go to tab **Properties and Graphs**.

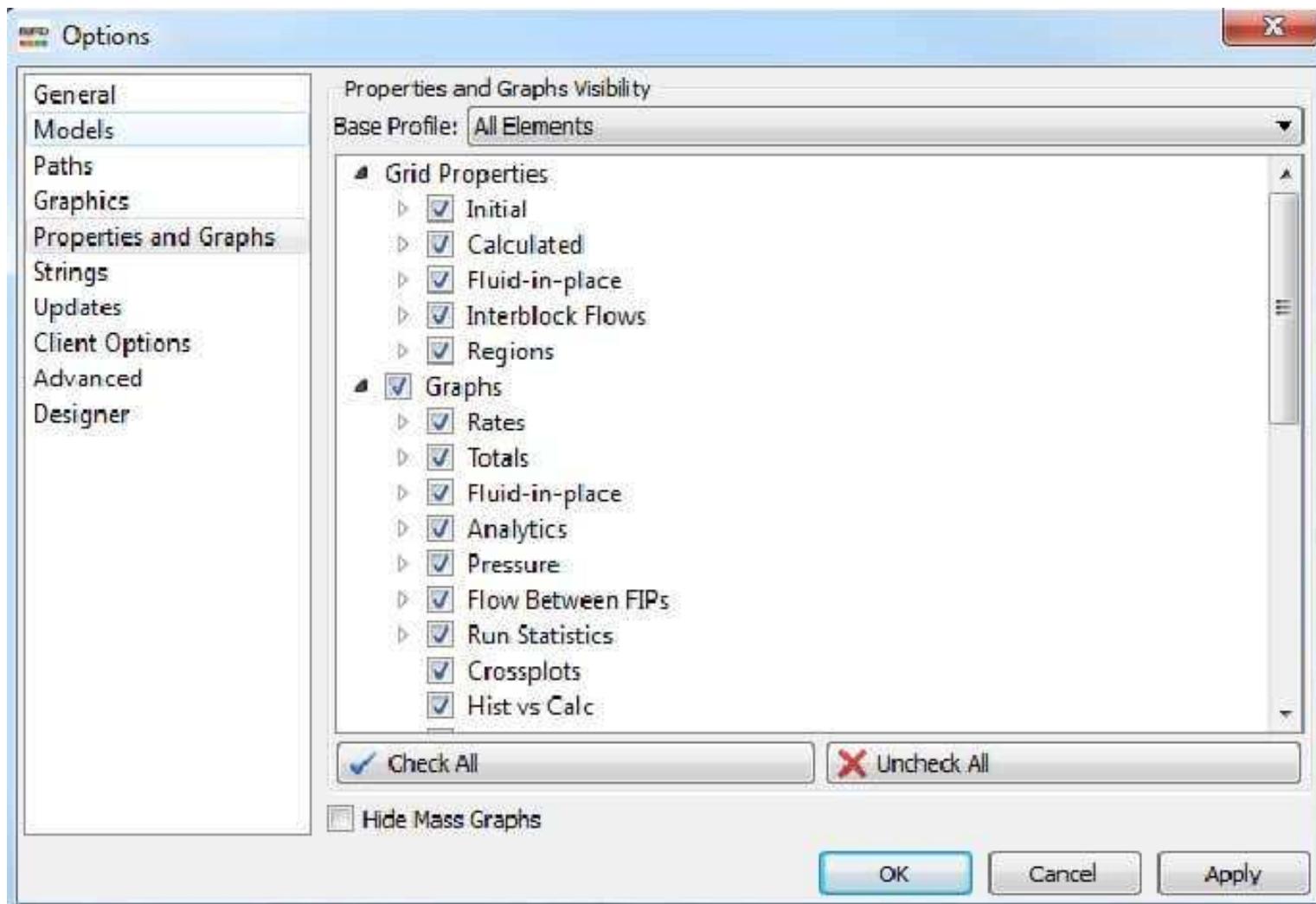


Figure 236. Dialog for selecting properties to be displayed and edited.

In this tab, you can select properties and graphs to be displayed:

1. Base Profile:

- All Elements (all possible properties and graphs will be displayed);
- Simple (this will display only certain properties and graphs, whose list can be viewed in the tree **Properties and Graphs**);
- Custom (the user creates a customized list of properties and graphs to be displayed by checking those required). On figure 237, for instance, **Initial** grid properties and **Interblock Flows** are unchecked. So those properties are not listed in the **Grid properties. Initial** option.

2. Hide Mass Graphs.

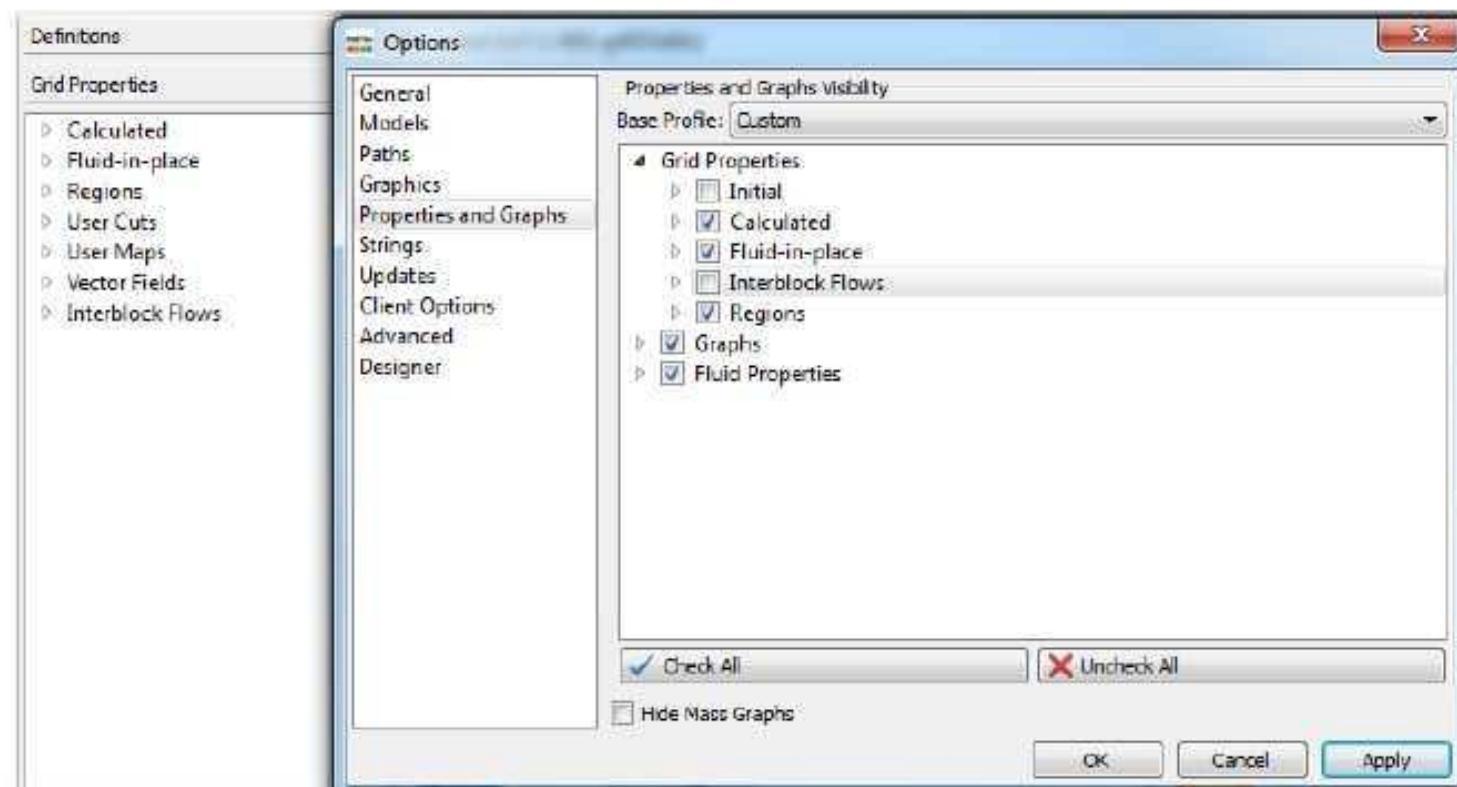


Figure 237. Initial properties and Interblock Flows are not shown.

17.6. Strings

To open the **Strings** dialog, go to tNavigator's main window, click **Settings**, and select **Options** in the pop-down menu. Go to the tab **Strings**.

Settings available:

- **Thousands, Millions, etc. Number Format;**
- **Show Small Values with Actual Precision** (not set by default);
- **Number Precision in Tables: Format** (floating-point format, for example 0,012, or exponential format – 1,2e-2) (see figure 238), **Maximum number of significant digits** (default – 6);
- **Precision of Palette Labels** (the number of digits after the decimal point – the default setting is 5 digits).

Grid Block Coordinates Representation.

Place the cursor on a block to see the following information displayed below in 2D or 3D view:

- Grid Block Number,
- Block Center Coordinates (in meters),
- Internal Grid Block Number (is not shown by default);

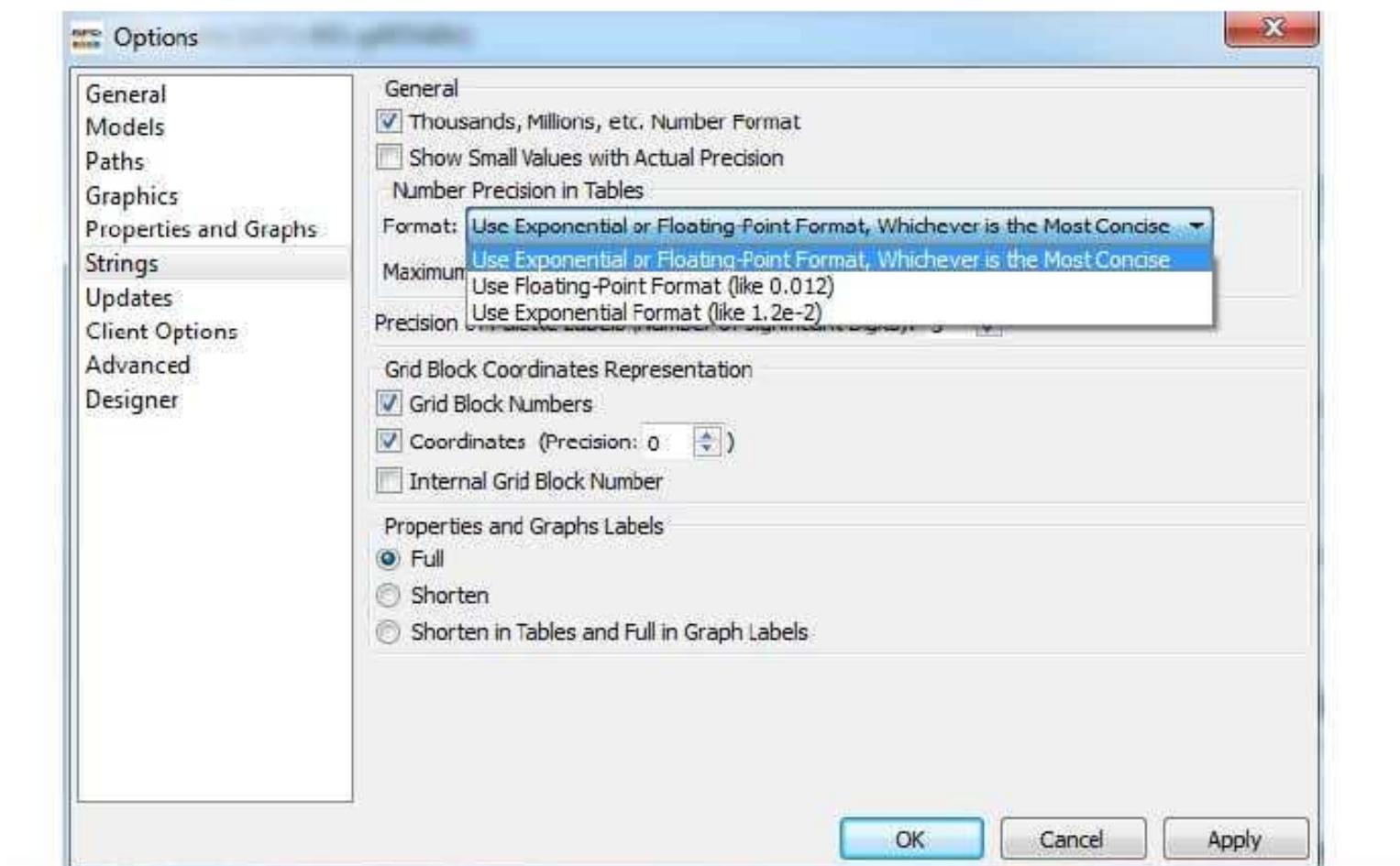


Figure 238. Format number in tables.

An example is in the figure 239:

Block size along X [22, 43, 1] = [-1264, 6110, 1747] = [5121] = 122.649471 m. This means that: for the block with the coordinates [22, 43, 1], the block center's coordinates in meters are: [3470, -1995, 1424], the block's internal number is: [31058], the block size is: 122.649471 m.

Maps' and graphs' names (labels):

- Full;
- Short (names will be visualized in the form PRES, SOIL, SWAT, WOPT, WOPTH etc.);
- Short in Tables and Full in Graph Labels.

Maps' and graphs' names are displayed full by default.

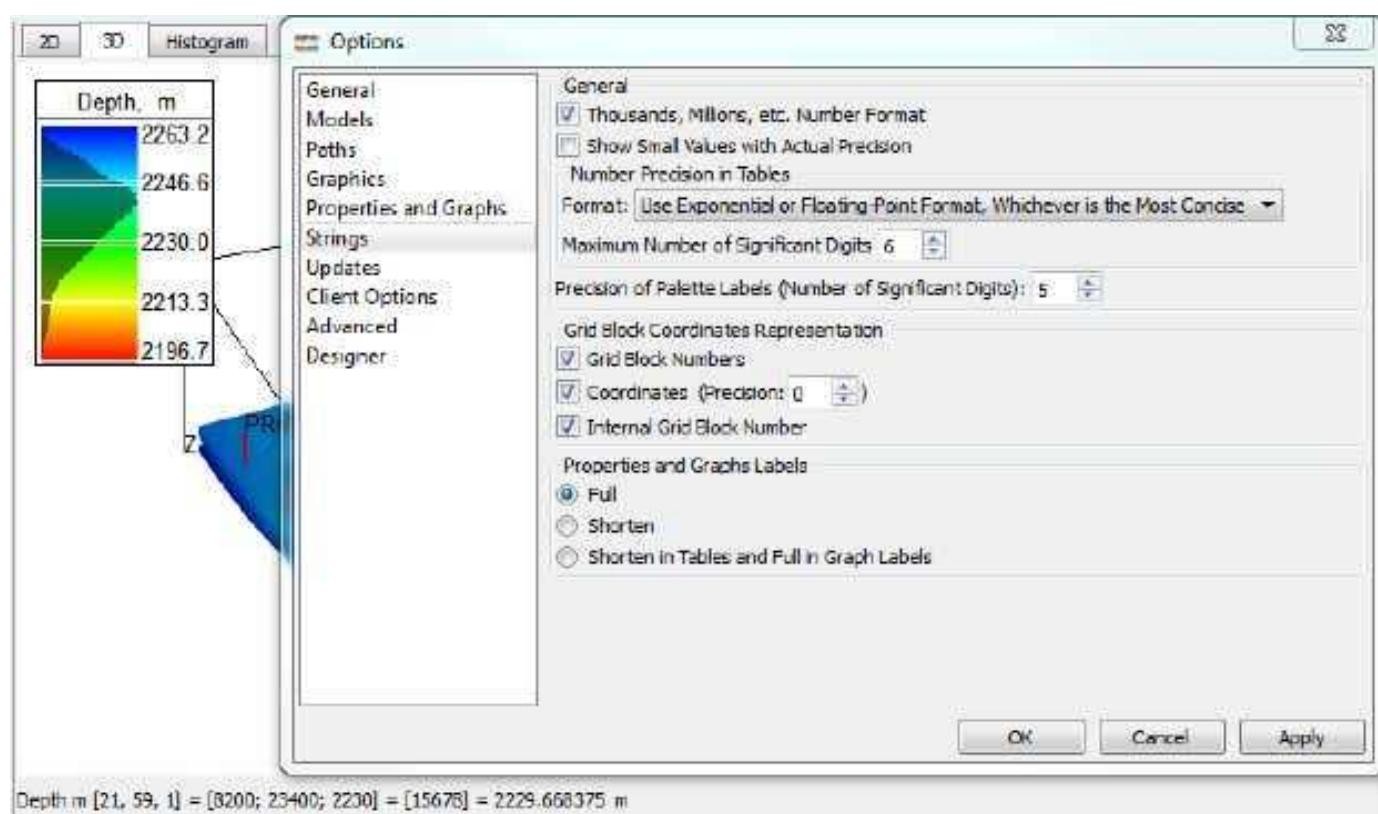


Figure 239. Map captions settings.

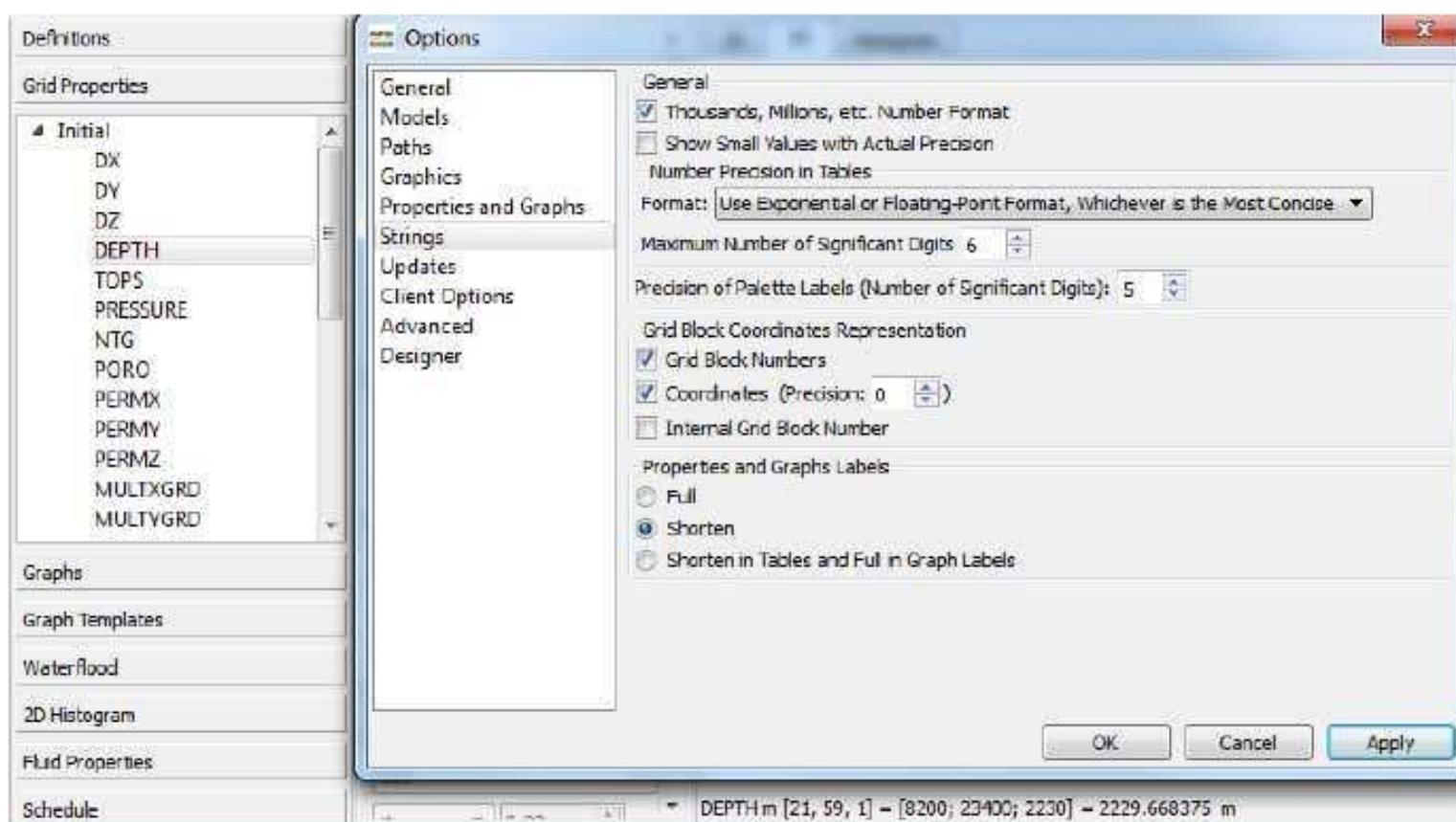


Figure 240. Maps' short names.

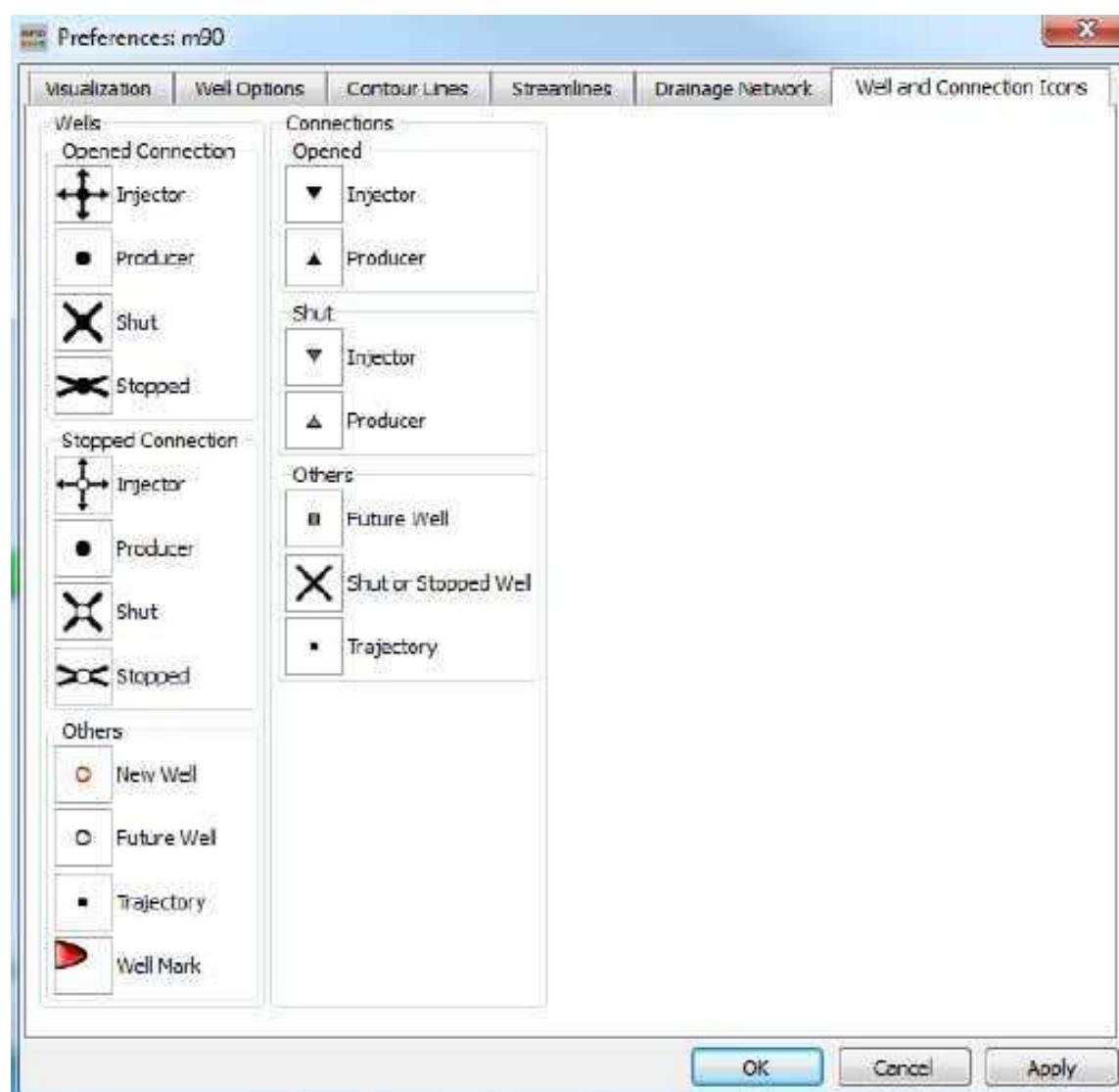


Figure 244. Preferences. Well And Connection Icons.

18. References

- [1] Nelder, J.A. and Mead, R., A simplex method for function minimization, *Comput. J.*, 7, pp. 308–313, 1965.
- [2] Kathrada, Muhammad, Uncertainty evaluation of reservoir simulation models using particle swarms and hierarchical clustering *Doctoral dissertation, Heriot-Watt University*, 2009.
- [3] N.S. Bahvalov, N.P. Zhidkov, G.M. Kobelkov, *Numerical methods*, M. «Nauka», 1987 [in russian]
- [4] Clayton V. Deutsch, *Geostatistical Reservoir Modeling*, Oxford University Press, 2002
- [5] A. Bardossy *Introduction to Geostatistics* University of Stuttgart
- [6] S. D. Conte, Carl de Boor *Elementary Numerical Analysis* McGraw-Hill Book Company, 1980.
- [7] J-P Chiles, P. Delfinder *Geostatistics Modeling Spatial Uncertainty* Wiley & Sons, Canada, 1999.