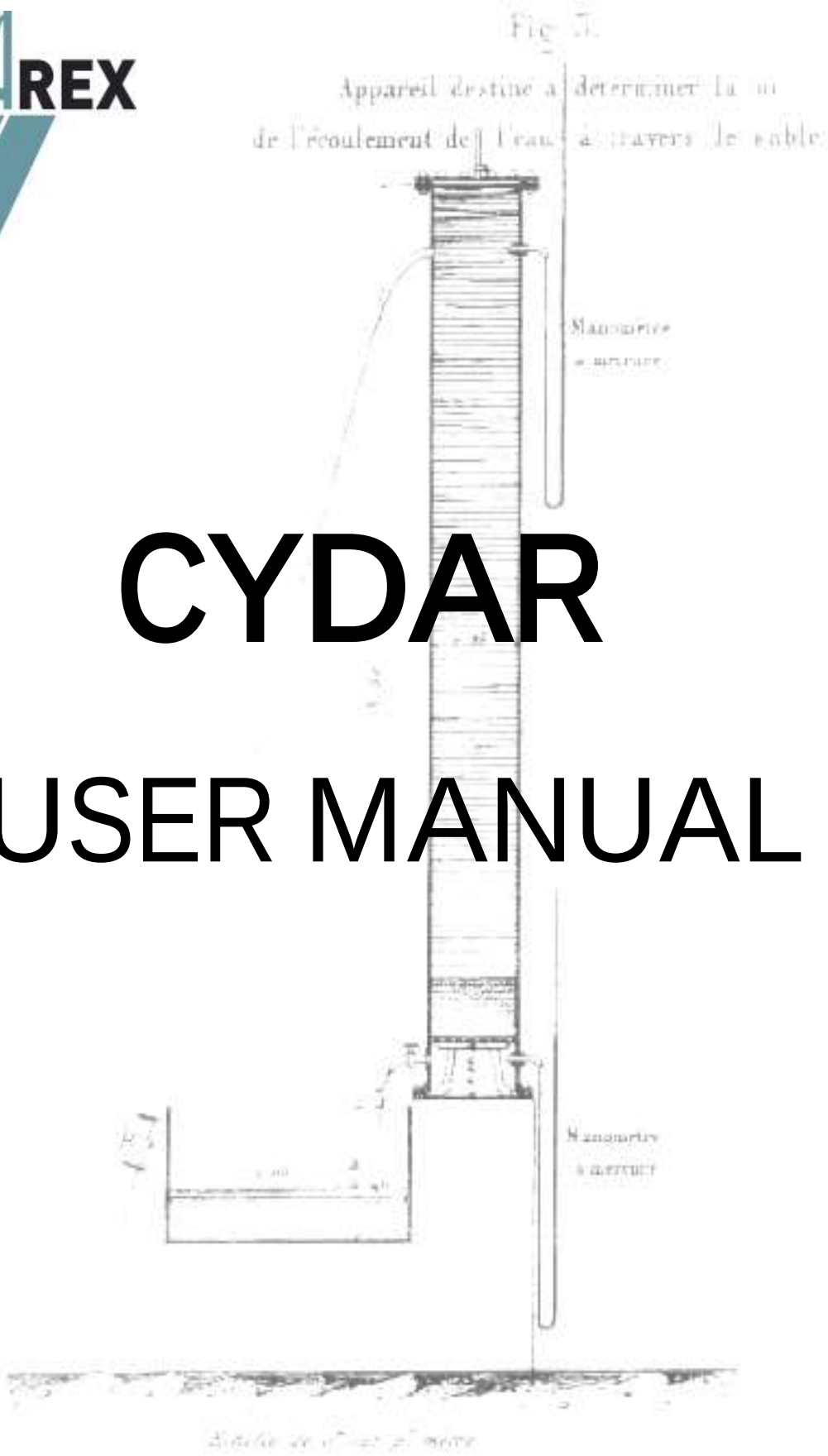
CYDAREX



CYDAR

USER MANUAL

CYDAR – USER MANUAL

This notice describes the functionalities of CYDAR. The first part details the installation procedure, gives an overview of CYDAR, and presents the main functionalities such as importing and exporting data, smoothing curves, displaying graphs. Each module is then presented. The relative permeability and centrifuge capillary pressure are part of the Two-Phase Flow module.

Other modules related to laboratory equipment with data acquisition boards (Darcylog, Darcypress, Centri) are described in separate manuals.

Installation of CYDAR

System requirements

CYDAR is developed in Visual Basic, an object oriented language that takes advantage of the Windows graphic environment. CYDAR runs on a Microsoft Windows operating system, and has been fully tested on Windows XP, Windows Vista, Windows 7, 8, and 10. CYDAR does not require a powerful computer, and can be installed on a laptop or an older computer. CYDAR does not require access to the Internet.

Numerical calculations such as the optimization loops are developed in FORTRAN using the powerful IMSL library. However, there is only one program to install, and the user has no direct contact with the various FORTRAN's DLL or other graphical objects used by CYDAR.

The use of data acquisition boards requires installation of the corresponding software. This operation is described in separate manual.

Installation of CYDAR

Before installation, it is recommended to close all running applications.

Insert the installation software CD in your CD-Rom drive. The CD contains other folders (tutorials, User Manuals, etc.). If the installation CD does not start automatically, open your CD-Rom drive and double-click on "setup.exe." For Windows Vista, Windows 7, 8, and 10, it is recommended to use the option "run as administrator", using the mouse right click (even if you use an administrator account). If CYDAR was downloaded from the Internet, just double-click on the Install file.

On Windows 8 or Windows 10, your computer might require an Internet connection to finish the installation. Also, if the installation blocks with the following message: "Microsoft data access components 2.0 (not responding)", kill the process when prompted and let the installation finished. The installation should be fine.

The installer will start. Click the "install" button and follow on-screen instructions.

Depending on the operating system, some messages might occur during installation. Choose the option "ignore" and continue. When prompted for files "not newer than the existing one," use the option to keep the existing file as recommended.

After the installation is completed, click "finish".

During the first utilization, a login and password will be required; contact CYDAREX if you do not have one. For registered clients, the login and password can be changed by opening the "about" menu in a blank project.

Problems with the Installation

For problems with the installation of CYDAR, see the Troubleshooting section at the end of this manual.

Updating CYDAR

Most updates can be installed by replacing the "cydar.exe" file by a newer version downloaded from CYDAREX website. The file to replace is usually located in C:\Program Files\CYDAR. Occasionally, a complete reinstallation of CYDAR might be required. In that case, it is recommended to first uninstall the software, and then perform a complete installation with the files downloaded from CYDAREX website.

Uninstalling CYDAR

To uninstall CYDAR, open "Add or Remove Programs" in Windows control panel. Then select CYDAR and click on "change/remove".

Silent Install and Uninstall of CYDAR

To install CYDAR in silent mode, type the following command line:

c:\CYDAR_Install_Folder_Path\setup.exe –s c:\ CYDAR_Install_Folder_Path\CYDAR_Install.log

where "CYDAR_Install_Folder_Path" corresponds to the path of CYDAR installation folder.

To uninstall CYDAR in silent mode, type:

St6unst -n "C:\Program Files\CYDAR\St6unst.log" -f -q

CYDAR Overview

CYDAR has been developed in collaboration with core analysis specialists, with two main objectives:

To be user-friendly: This is achieved by using Windows environment with an intuitive graphic user interface, like any Microsoft software. CYDAR can be used with no specific knowledge in numerical simulation or reservoir engineering.

To be accurate and powerful: CYDAR uses the most recent methods developed and tested in research laboratories, and published in the proceedings of the Society of Core Analysis.

CYDAR contains the following modules for routine and special core analyses:

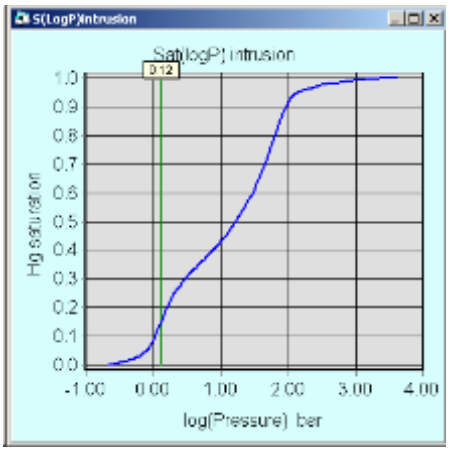
- mercury injection and withdrawal (MICP),
- absolute permeability,
- two-phase flow experiments,
- steady-state and unsteady-state relative permeability,
- centrifuge capillary pressure and relative permeability.

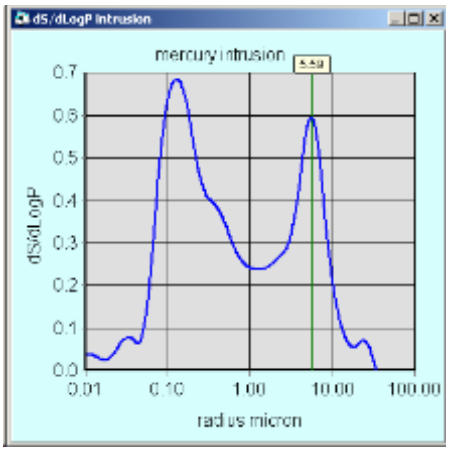
Mercury Injection (MICP)

Mercury injection is used to calculate drainage capillary pressure for initializing reservoir simulations (J Leverett function) and to determine pore size distribution (EOR and rock typing). Entering data takes just a few minutes and computation is instantaneous.

Main features include:

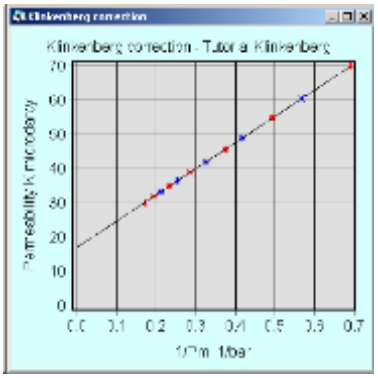
- ASCII files generated by experimental apparatus can be opened directly,
- powerful data smoothing using spline functions,
- calculation of three types of pore size distributions,
- J Leverett functions, reservoir transition zone, and water cut,
- estimation of permeability (Swanson and other correlations),
- automatic reporting: all results are exported in an Excel file.





Absolute Permeability

In laboratories, absolute permeability is often calculated with a spreadsheet in Microsoft Excel. However, the use of CYDAR minimizes the risk of errors, improves quality control, and calculations are performed numerically without approximations.



Main features include:

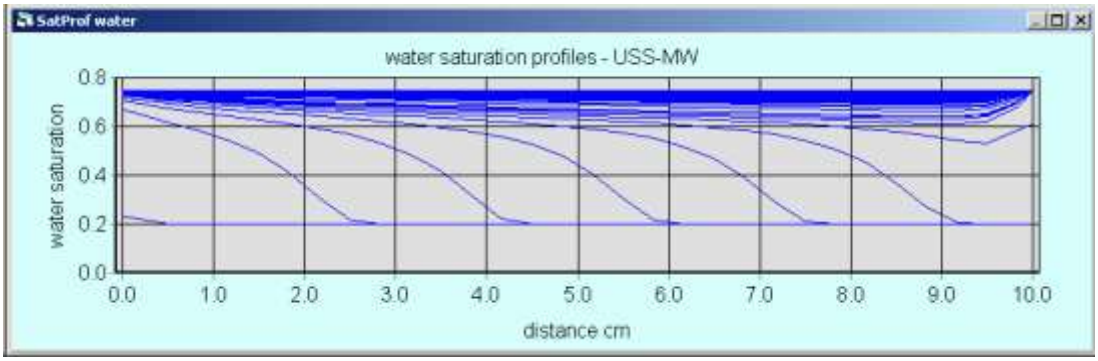
- gas and liquid permeability, steady-state and pulse decay,
- determination of the inertial coefficient (Forchheimer correction),
- determination of the Klinkenberg correction,
- correction for pressure drop in tubings,
- automatic reporting: all results are exported in an Excel file.

Two-Phase Flow Experiment

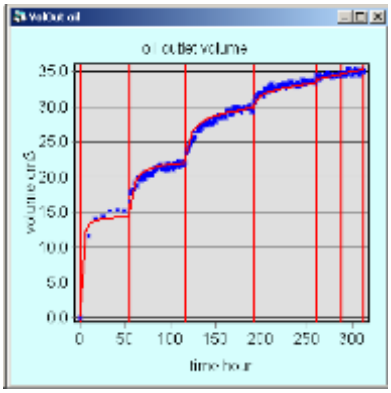
CYDAR can simulate most of two-phase flow experiments such as:

- relative permeability in steady-state and unsteady-state,
- semi-dynamic method,
- spontaneous displacements (immersion of a sample into a fluid in drainage or imbibition),
- gravity flow,
- centrifuge displacements in drainage or imbibition,
- porous plate, with pressure drop inside the porous plates.

Simulations are performed on a one-dimensional sample, either homogeneous or with heterogeneities. Heterogeneities include permeability and porosity continuous profiles, or composite samples. Hysteresis between drainage and imbibition is also taken into account.



Most laboratory experiments are preset in CYDAR, and boundary conditions, which are specific to each experiment, are adjusted accordingly. In addition, non-programmed experiments can be simulated by setting inlet and outlet boundary conditions.

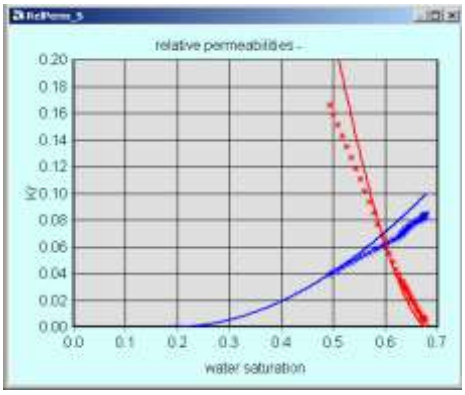


The two-phase flow module can be used to design an experiment when most properties of the sample are known or estimated, or to determine parameters from measurements (inverse calculation). For all experiments, Kr and Pc curves can be adjusted by manual or automatic history matching. The simulated results (production, saturation profile, and pressures) are compared to experimental data; the difference is minimized in an optimization loop.

Relative Permeability

Determination of relative permeabilities Kr is one of the main objectives of special core analysis. It is now well recognized that numerical interpretations accounting for capillary effects are absolutely necessary.

Relative permeabilities can be determined by history matching of any transient experiment in steady-state (SS) and unsteady-state (USS) displacements, centrifuge, and semidynamic methods.



Main features include:

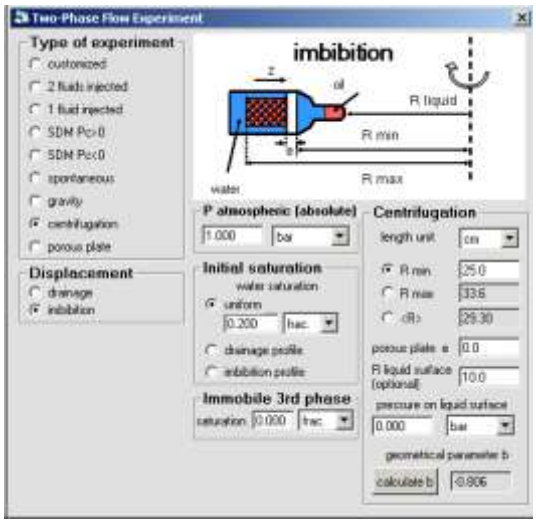
- Kr models include Corey, modified Corey, and LET functions.
- for unsteady-state displacements, analytical calculation using JBN, or Jones and Roszelles methods,
- for steady-state displacements, analytical calculation assuming uniform profiles,
- for SS and USS displacements, numerical simulation with capillary pressure, and determination of Kr using manual or automatic optimization (so-called history matching),
- automatic reporting: all results are exported in an Excel file.

Centrifuge capillary pressure

The centrifuge module converts the average saturation measured during the experiment to local saturation at the entrance of the sample, and additionally allows the evaluation of relative permeabilities.

Main features include:

- displacement in drainage and imbibition,
- choice of several interpretation methods such as Hassler Brunner, Forbes, and spline functions,
- relative permeabilities determined by Hagoort method or multistep history matching,
- automatic reporting: all results are exported in an Excel file.



CYDAR main features

All modules in CYDAR share the same features for data processing, graph and table displaying, and exporting results.

Windows environment

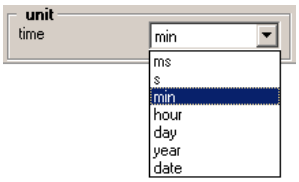
CYDAR is developed in Visual Basic, an object oriented language, which takes advantage of the familiar Windows environment. Some numerical calculations such as optimization loops use FORTRAN and its powerful IMSL library. However, all FORTRAN DLLs are controlled by CYDAR, and the user interacts only with the graphic user interface.

CYDAR does not require a high-performance computer, and runs perfectly on a laptop computer. CYDAR can be installed on any Windows operating systems including Windows XP, Vista, 7, 8, and 10, and does not require an Internet connection.

Data input

Several methods are used to input data. A small amount of data can be directly typed in; larger files can be cut and paste from Microsoft Excel or other applications. Some ASCII files generated by experimental apparatus can also be opened directly (like Autopore for mercury injection). CYDAR can also acquire data during experiments by reading pressure, temperature, and other parameters via USB connections.

For each parameter, the unit can be chosen in input, output, or for graph display.



Numerical calculation

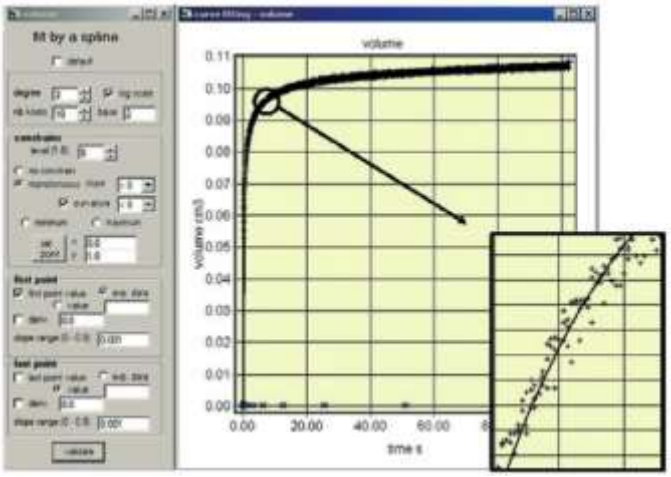
Simulations are performed on a 1-D grid. The size of the grid and parameters for the numerical simulation can be adjusted for speed or accuracy. During numerical simulation and optimization, all variables (flow rates, effluent volumes, pressures, saturations...) are displayed dynamically; and the user can stop the simulation at any time to change parameters.

Exporting results

Graphs can be printed or copied into the clipboard in metafile or bitmap formats, and can be cut and paste into a Microsoft Word or Excel report. Graphs can also be saved as metafile, bitmap, or JPEG files. For each graph, the corresponding data set can be displayed as a table and copied into Microsoft Excel or other software by a cut and paste.

Data smoothing

Most experimental data are noisy and an efficient smoothing method is required for computation or comparison with simulated results for history matching. CYDAR offers several analytical functions for smoothing such as exponential, power law, and spline functions, the most general and powerful tool available.



Graphical display

Variables can be displayed as a graph at the end or during simulations. Graph parameters (scale units, log scale, legends, symbols, colours...) can be adjusted.

Reporting

All results can be exported as an Excel file. Pre-formatted reports are available, and can be easily tailored by the user. The company logo can be added to the reports.

CYDAR functionalities

This section describes some functions common to all modules and related to data handling and the graphic user interface.

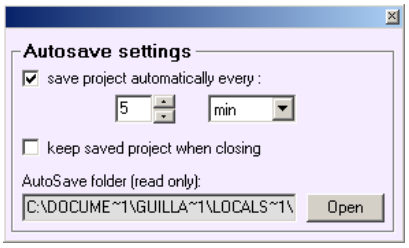
Opening and saving projects

A project can be saved by choosing "save" or "save as" in the menu "file". The extension ".cyd" is automatically added at the end of the project name.

Existing projects are opened following the standard Windows procedure, by double-clicking on the file. When opening a file for the first time, Windows may ask the user to select the appropriate program to connect the extension ".cyd" as a CYDAR file. Use the navigator to find CYDAR.exe in the program files folder.

AutoSave Settings

By default, a project is save every 10 min, but the delay can be reduced down to 1 minute. In order to avoid a crash during saving, the previous saved project is always kept. Consequently, there are always two saved projects with names xxxx_1 and xxxx_2. The saved project files can be saved when closing CYDAR. The user will be prompt to save a project when closing the application. To access the AutoSave settings, select **"Edit menu – settings AutoSave"**.



The directory in which the project is saved is the system temporary folder and cannot be modified. The **"Open"** button will open the temporary folder in Windows Explorer (icon in the task bar, not on the desktop)

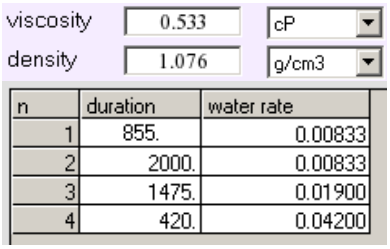
Useful Tips:
If CYDAR quits unexpectedly, you might recover some of the past work by opening the temporary AutoSaved file. Do not re-open the saved file, as it will erase the AutoSaved file. Instead, open a new CYDAR file and press **"open"** in AutoSave settings, and look for the most recent .cyd file.

Importing Data

Several methods are used to input data:

- Entry boxes: values are typed and validated by pressing the return key on the keyboard;
- Data spreadsheets: "copy and paste" from Microsoft Excel or other applications can be used.

CYDAR, with the module DAQ (Data Acquisition) can also monitor data during experiments by reading pressure, temperature, etc, via USB or RS232 connections (tailored applications like DARCYLOG™).



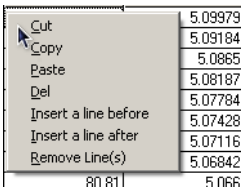
Useful Tips:
Please avoid quotation mark "" when entering text data (such as reference of samples, comments). This symbol is used as a text separator and the text after the first quotation mark will be ignored.
Shadowed text boxes are for display of results only and cannot be changed by the user.

Reading ASCII files

If required, CYDAR can open specific data files used by some equipments, such as the ".rpt" files provided by Autopore Mercury Injection device from Micromeritics Instrument Corporation. CYDAR can input data in the required format and units. An example is provided in MICP tutorials.

Loading data in CYDAR

Most experimental data can be imported clicking on **"Load Data"**, **"Data Points"** or **"Load/Fit Data"** (depending on the module). This opens a spreadsheet. Lines can be removed or inserted through the menu **"edit"** or the right click menu:



The left part of the spreadsheet has three frames:

- Load raw data: the user can select the rows interval, data type, and units.
- Noise filter: users can reduce noise by averaging data points over a specified number of points. When calculating average, the number of points furthest away from the average (end-points) can be removed. When using filtering, only filtered data are kept as raw data.
- File reduction: users can reduce the number of data points by removing data points when a variable remains constant.

The specificities of the spreadsheets with the modules are described in the corresponding paragraphs.

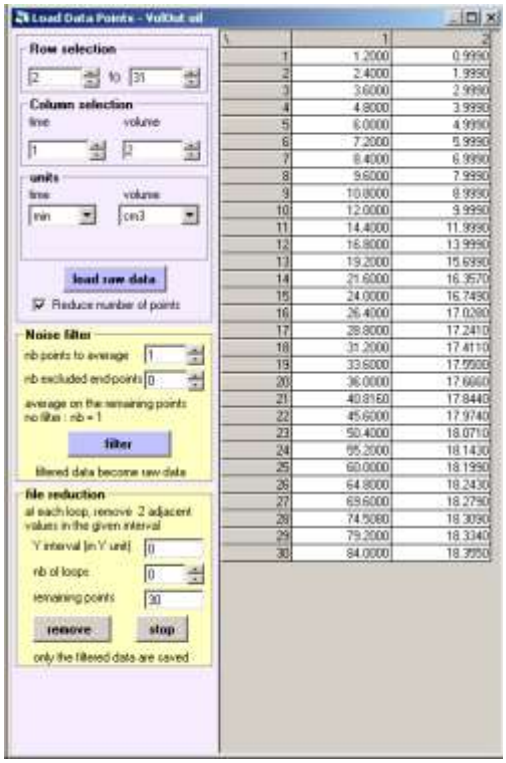
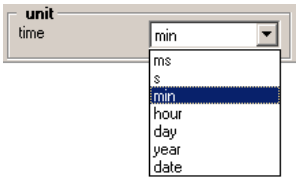


Figure 1: a spreadsheet in CYDAR. Data are loaded by clicking on "Load Data".

Units

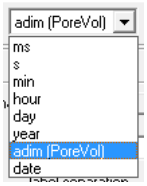
All common units can be selected in combo boxes. Each data can have its own units: for instance, a sample can have its diameter expressed in inches and its length in cm. For calculation, all values are converted into the International Unit System, and results can be displayed in the selected unit.



For volume units, the Pore Volume unit ("PoreVol") is a dimensionless unit corresponding to the volume divided by the pore volume.

For time units, the "adim (PoreVol)" unit is a dimensionless unit defined as:
$t^* = \frac{V_{injected}(t)}{V_{pore}}$, where t^* is the dimensionless time, $V_{injected}(t)$ the total volume injected at time t, and V_{pore} the

pore volume. The “adim (PoreVol)” will only appear if it can be calculated. The dimensionless time is available for most experiment types in the TPF module.

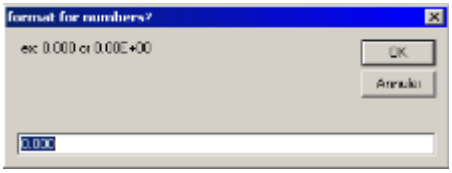


Note: If “adim (PoreVol)” is not available, data for the total volume for inlet sample need to be entered: once a simulation is run and the total volume for inlet sample has been calculated, right click on the total volume graph, and select “simulation to data points”. Data points for the total inlet volume are now available, and “adim (PoreVol)” should be a time unit option.

Number format

To change the number format in a box, place the cursor on the number and select the "format" menu (or press Ctrl+F). In the displayed window, enter the format following Microsoft standard. For example:

- 0 for an integer (ex: 1343)
- 0.00 for a decimal number with a given number of digits (ex: 1343.45)
- 0.00E+0 for scientific format with two digits after the decimal point and 1 digit in the exponent (ex: 1343.45E+0)
- mm/dd/yy hh:nn:ss for date (ex: 08/25/10 12:02:36).



Data editing

The "edit points" button opens the following window. The editing panel presents the following functions:

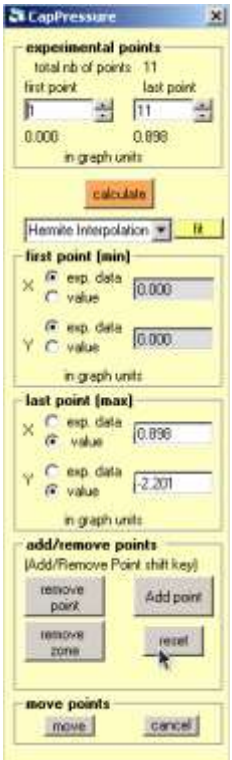


Figure 2: Edit points panel.

experimental points: selection of a given interval of data points. The total number of points is shown and the user can select the first and last points used to fit or interpolate experimental data. The corresponding abscissas are displayed in graph units.

first and last points: end points coordinates in graph units. Both X and Y can be selected to be equal to the end points defined above or can be imposed.

add/remove points: To add a point click on "add point" button, the shape cursor changes to "+", then hold down the shift key and left click. A cross symbol × will represent the added point (by default, the added points are visible).

To remove a point, select "remove point", hold down the shift then left click on the point to be removed.

To remove an entire zone of points select "remove zone". There are two options: one is to move the cursor over the points to be removed holding down the shift key; the second is to select the zone with the mouse (top left to bottom right, holding down the left button) hold down the control key then release the mouse button. All modifications can be cancelled using the "reset" button.

move points: when the “move” button is clicked a data table is displayed. There are two ways to move a point: changing the values in the table or moving a point on the graph. On the graph, the mouse pointer will change to the hand cursor over movable points. To move a point left click on it and drag-and-drop. To validate any move, the “calculate” button must be pushed. All moves can be cancelled using the "cancel" button.

Calculate button: to actualize any change, or to end move/add/remove tool, “calculate” must be pressed.

Remark: The “reset” in the add/remove section will erase any modifications.

Undo-Redo: there is a five-step undo stack. “reset” will erase the stack. Cancelling the “move” tool set back the stack to its previous state. The undo/redo shortcuts are respectively “Ctrl+Z” and “Ctrl+Y”.

Data smoothing

Most experimental data are noisy and an efficient smoothing is needed for computation or history matching. Several analytical functions are available for smoothing, such as exponential or power law functions, but the most general and powerful tool is the spline functions.

Principle of data smoothing

Following a purpose of quality control, experimental data cannot be modified when they have been entered in CYDAR. However, the user can decide not to account for some experimental points which can be obviously erroneous. The user may also decide to add points needed for numerical simulations, such as end-points in relative permeability. The operation of adding and removing points is called "editing" and is accessible in most modules by selecting the "Edit Points" button. Figure 3 shows an example where two end-points are added, at x=0 and x=1, and three points removed from the raw data. The "edit points" function is detailed below.

An analytical function is then calculated to represent a continuous form of the discrete experimental values. The default case is a linear fit, which is a linear interpolation between each data point, represented as a pink curve on Figure 3. This analytical function is displayed with a number of points (100 by default) which can be changed in the "Graph edition window".

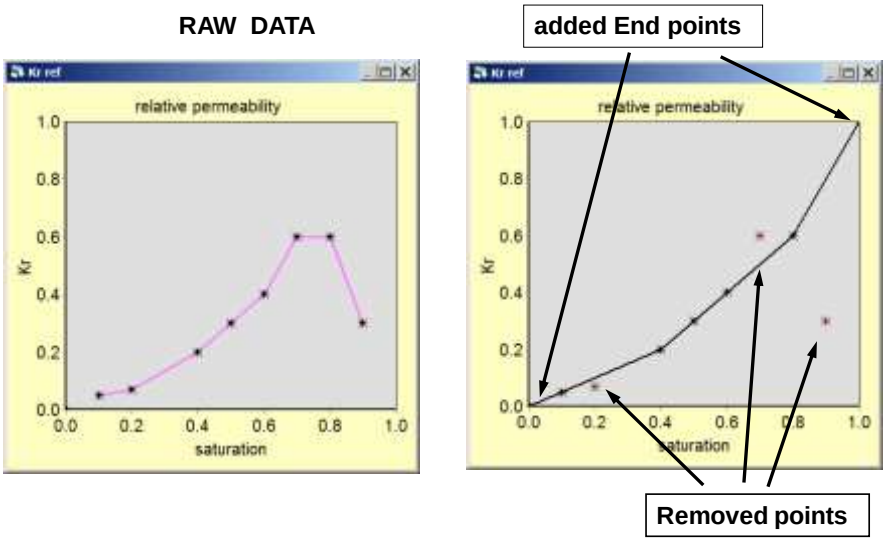


Figure 3: The pink solid line represents the analytical fit with a linear interpolation between all raw data points. The solid black line represents a linear interpolation between consecutive points, with two end-points added and 3 data points removed.

Curve fitting

CYDAR provides many analytical functions for smoothing and interpolation: "linear", "splines", "power", "modified Power", "LET", "log(X^beta)", "bi-exponential", "biExp Multi-Step", "parabolic", "modified hyperbolic", "Splines Multi-Step", "Splines Interpolation", "Akima Interpolation", and "Hermite Interpolation" (Figure 4).

These interpolation functions need to be used when optimizing Kr by points.

Figure 4: Various functions available for fitting.

Interpolations (curve passes by all data points)

- Linear: This corresponds to a linear interpolation between each data point.
- Splines Interpolation: cubic spline interpolation with zero 2nd derivatives at end-points.
- Akima Interpolation: cubic spline interpolation using Akima method to avoid wiggles in the interpolant.

- Hermite Interpolation: Hermite cubic spline.

Spline Fits (curve does not pass by all data points)

A spline fit is the most powerful fitting tool available, providing several constraint controls (Figure 6). The user can specify the spline degree and the number of knots. The default value is 3 for both. Knots can be placed on a linear scale (default) or on a log scale.

Briefly, a spline fit is a piecewise polynomial fit (of the specified degree). Data between each knot are fitted with a polynomial. At each knot, there is continuity of the curve, and of the first derivative.

> **Useful tip:** For a better fit, increase the number of knots, not the spline degree.

Constraints can be imposed on the slope and curvature of the fit ("Constraints" frame in Figure 6). For example, Kr curves are assumed to be monotonous in a homogeneous sample.

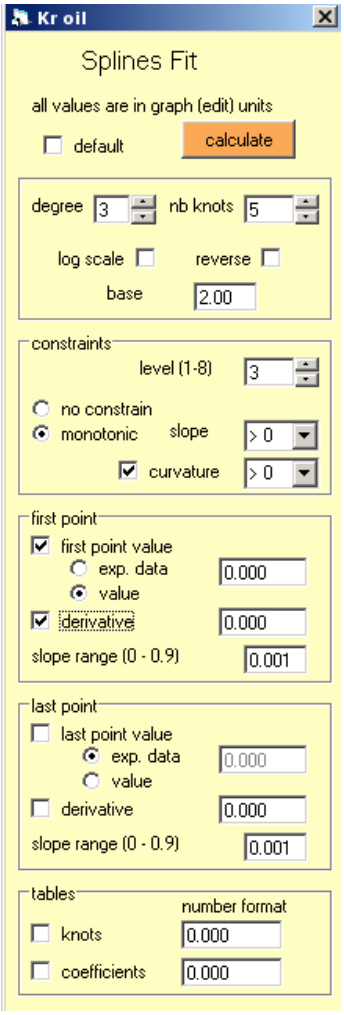


Figure 6: Fitting with spline functions.

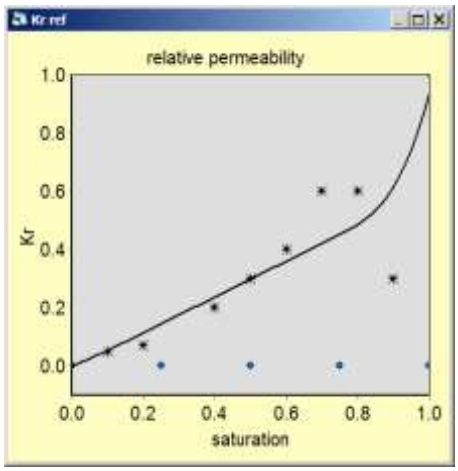


Figure 5: Fit of the data shown in Figure 3 using spline functions. Black line is fit with the constraints; blue circles are knots.

Two frames are used to impose the Y-coordinate of the end-points of an experimental or a given value, as well as its derivative (Figure 6). For instance, this function could be used to impose end-points of the Kr.

The tables section allows the display of splines knots and coefficients in tables with a given number format.

Fit functions

Other fits are available (Table 1), and the analytical equation is usually displayed in the fit window. Some functions are used for specific applications:

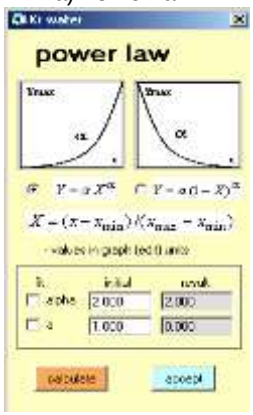
- The power function is used for calculating the "Corey" fit of the relative permeabilities.

- The "modified Corey", or modified power law, is used to fit relative permeabilities, and corresponds to the equation:

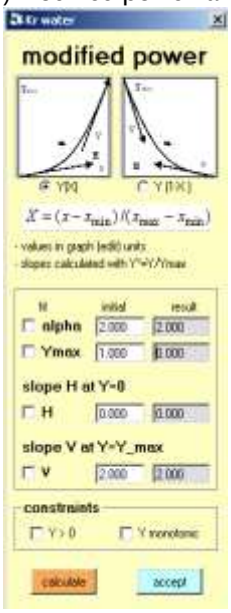
$$Y = Y_{\max}\left(\frac{a}{2a}x^{2a} + \frac{b}{a}x^{a} + Hx\right)$$ with a = V-b-H and b = 2α(1-H)-V+H.

- The LET function is described in Reference 4, "A new versatile relative permeability correlation", Lomeland et al., SCA, 2005, and is usually used for relative permeabilities.
- The hyperbolic function is used for calculating the permeabilities from mercury injection by the Thomeer method[13].

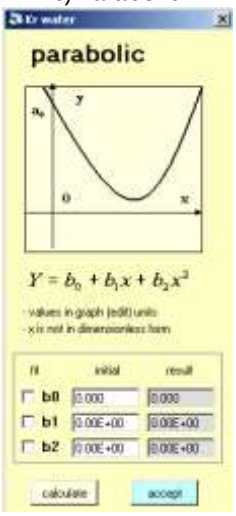
a) Power law



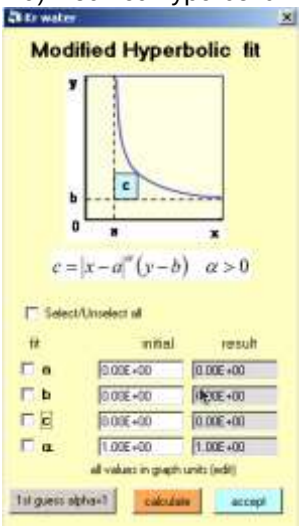
b) Modified power law



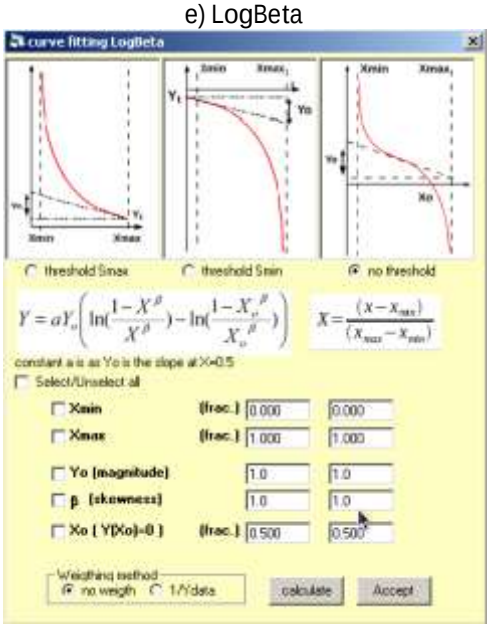
c) Parabolic



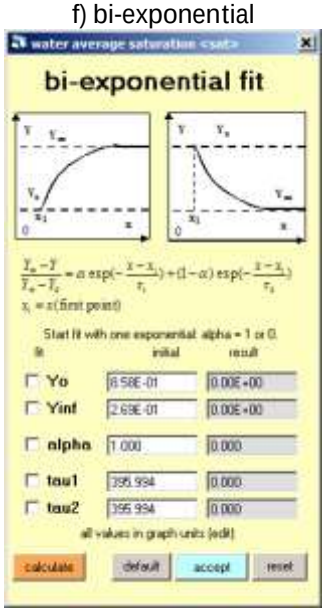
d) Modified hyperbolic



e) LogBeta



f) bi-exponential



g) LET function

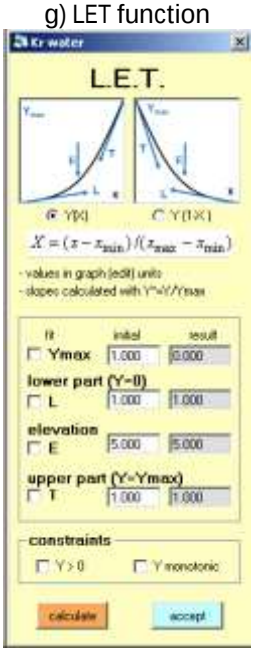


Table 1: numerous fits are available.

Multi-step fits

These fits can be used only in two-phase flow cases, when a specific variable called “block times” can be defined (see description of Two Phase Flow simulations).

Multi-Step Splines (Figure 7) allows using the CYDAR spline fit tool on each block time separately. The spline settings can be applied globally or per block. If “per block” is selected, the user must choose the blocks on the table by clicking on the left column. The corresponding area is highlighted on the graph with a yellow band as shown on Figure 8.

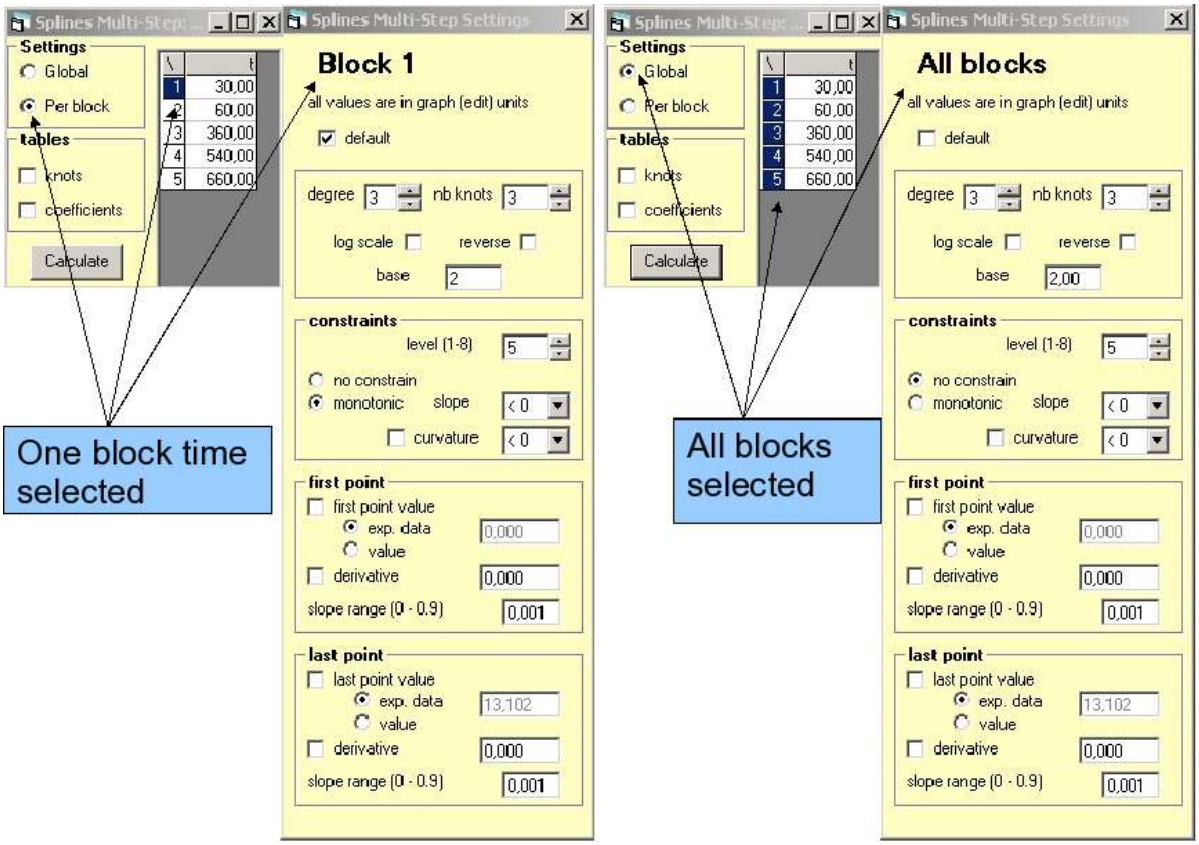


Figure 7: Multi-step splines fit

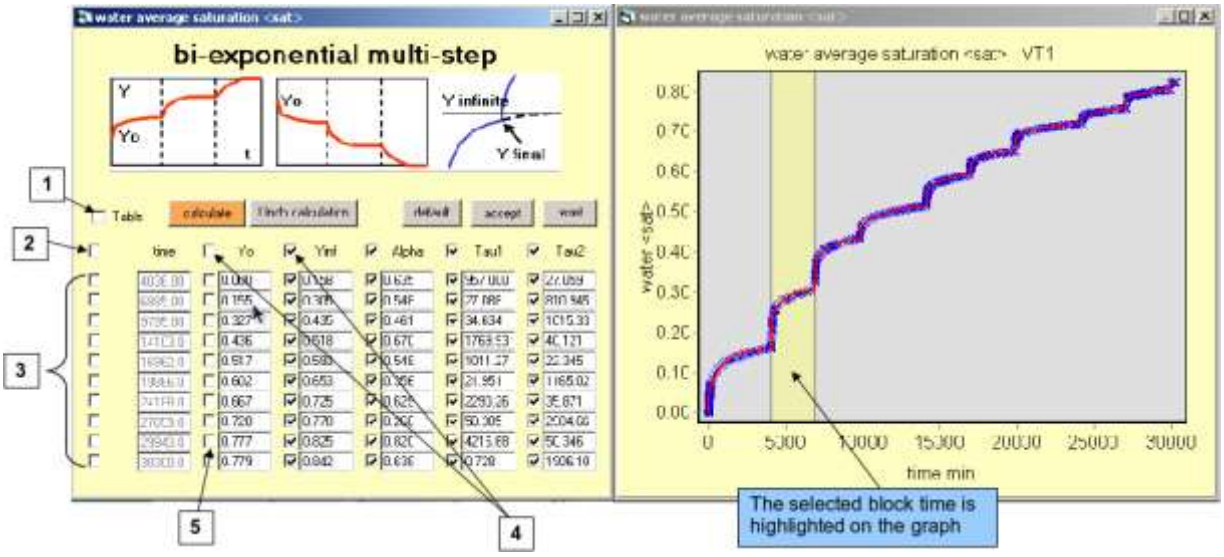


Figure 8: Multi-Step bi-exponential fit.

Multi-Step bi-exponential fit uses the bi-exponential fit (Figure 8) on the selected block times.

Check boxes (numbered 1 to 5 on Figure 8):

1. Open a table with the parameters allowing the user to copy and paste the parameters to another application.
2. Select/unselect all five parameters on all block times: Yo, Yinf, Alpha, Tau1 and Tau2.
3. Select/unselect a set of five parameters on one block time.
4. Select/unselect one parameter on all block times.
5. Select/unselect one parameter on one block time.

Buttons (Figure 8):

- "**calculate**": calculation of parameter. If no parameters are selected, the curve will be updated.
- "**undo calculation**": reset parameters to their values before last calculation.
- "**default**": Y_o and Y_{inf} from data points, Alpha = 0.5, Tau_1 and Tau_2 are the third of the block time interval.
- "**accept**": save the set of parameters.
- "**reset**": reset parameters to their values at the beginning of the fitting session.

Graphs

CYDAR uses two types of graphs:

- "Edit Points" graphs are used for editing data points (either experimental or calculated) with a default yellow background. These graphs are automatically open when calling editing or fitting functions.
- "XY-graphs" are for displaying input or output data; they have a default blue background. These graphs can be opened with the "view" menu of the main Window.

Edit Points graph

- Filtered data are called "raw data" in CYDAR (Figure 9).
- Some corrections can be performed (for instance, calculation of offset and dead volume in the two phase flow module) leading to "corrected data".
- As presented previously, the edit function is used to add, remove, or move points, and such editing leads to "edited points".
- The "edited points" are used to calculate fits, called "analytical function" in CYDAR. Those functions are then used for analytical calculations and interpolation.

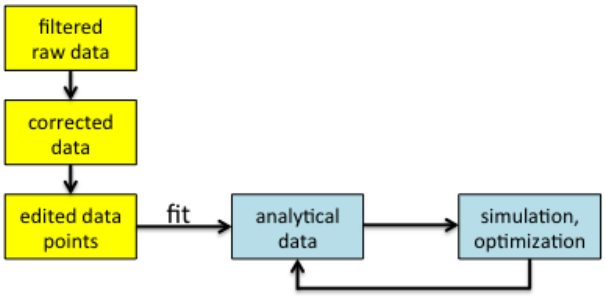


Figure 9: Flow chart for the raw data. A yellow background is used for Edit functions and graphs. A blue background is used otherwise.

XY Graph

Experimental values are compared to simulation results using the XY graphs, with a blue background. This graph displays:

- edited points
- analytical fit (points and line)
- results of numerical simulations
- a backup of simulations, kept in memory for comparison.

The two kinds of graphs, edit points and XY have the same following functions:

Scaling up and scaling down a graph selection

To scale up a part of graph, select the corresponding section with the mouse (top left to bottom right) holding down the left button (Figure 10a; result shown in Figure 10b). To undo, select a section of the graph with no data, holding down the left button from the bottom right to the top left (Figure 10b; result shown in Figure 10c).

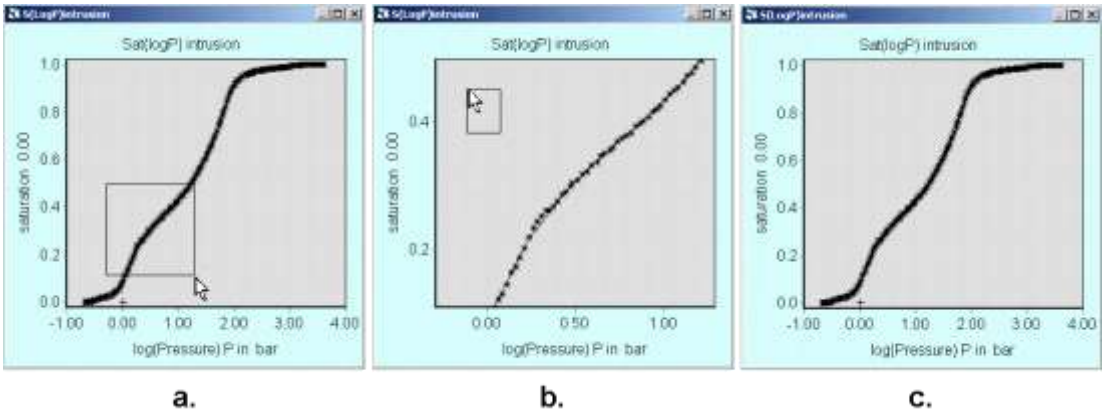


Figure 10: Zooming and unzooming on a graph window.

Moving the display zone

On a graph window, the user can move the display zone by pointing on the graph, holding down the right button, and moving the pointer. To go back to the normal view, the procedure is the same than to unzoom.

Editing a graph

Graph display parameters can be changed by double-clicking on the graph window or clicking on the axes. The Graph Edition panel (Figure 11) allows modification of most parameters, such as scale units, log scale, legends, symbols, colors, etc...

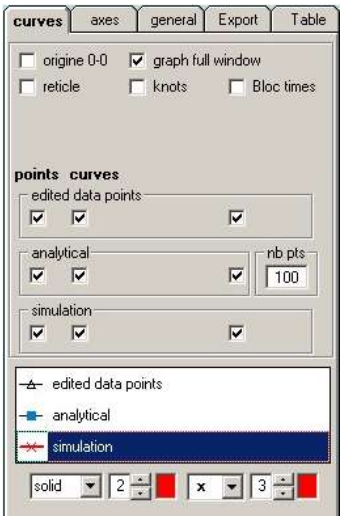


Figure 11: Graph edition panel.

Reticles

A reticle (a vertical moving line) can be used to display corresponding points between different graphs, for instance pore radius and capillary pressure (Figure 12).

To show the reticle, open "view" menu, and select "show all reticles". A green vertical line with a small box at the top appears on graphs. To move the reticle, click on one of the graph and press the arrow keys. A faster displacement is obtained by pressing simultaneously Shift and the arrows key.

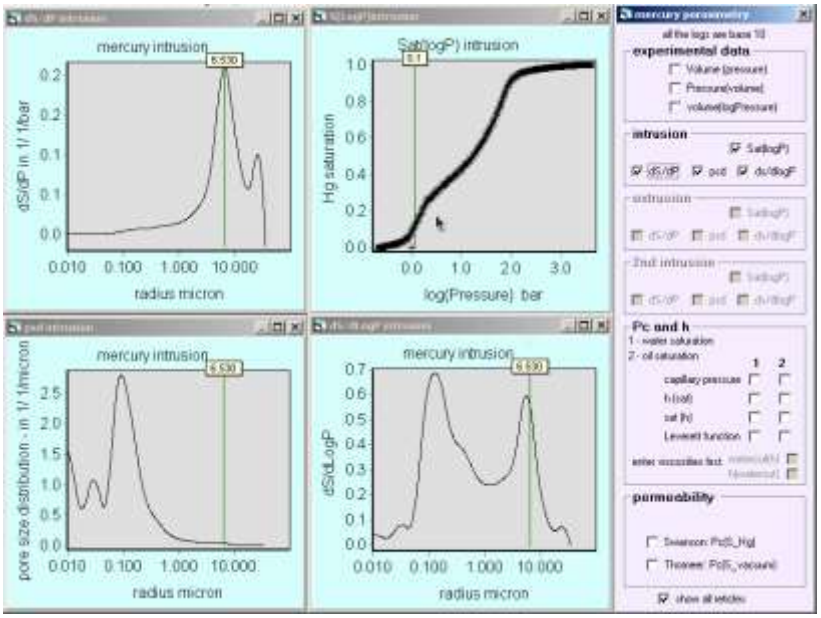


Figure 12: The reticle is the green vertical line showing the corresponding points between several graphs.

Exporting results

Graph

From the "export" tab of the graph edition panel (Figure 11), graphs can be printed or copied directly into the clipboard as a metafile or bitmap format. This format allows a cut and paste into a Microsoft Word or Excel document. Graphs can also be saved as metafile, bitmap, or JPEG files.

Simulation and analytical data

For any graph, data points can be displayed in a table using the "Table" tab in the graph edition panel (Figure 11). Data can then be copied and pasted into another software such as Microsoft Excel.

Reporting

Project can be exported as an ASCII format which can be directly opened with Microsoft Excel or a text editor. The report summarizes all data:

- General information: date, client name, operator, etc…
- Sample data: length, diameter, etc…
- Experiment data: type, set up, etc…
- Fluids properties: viscosity, density, etc…
- Simulation parameters: accuracy, mesh parameters, etc…
- Experimental raw data, edited data, analytical function, simulation data
- Tow-phase flow data:
- Analytical functions used for capillary pressure and relative permeabilities.
- Block time values
- Permeability data:
- Klinkenberg correlations parameters
- Mercury injection data

Useful tip: All data are exported in the same unit as their corresponding graph.

The report provides an easy editing format which, used with templates, allows automated and complex layout of all results. Figure 13 shows a part of an ASCII report for a steady-state imbibition. Figure 14 shows an example of layout using Microsoft Excel template with company logo, graphics, etc.

Pre-formatted reports can be available with the company logo and simple changes in the report are included in the CYDAR.

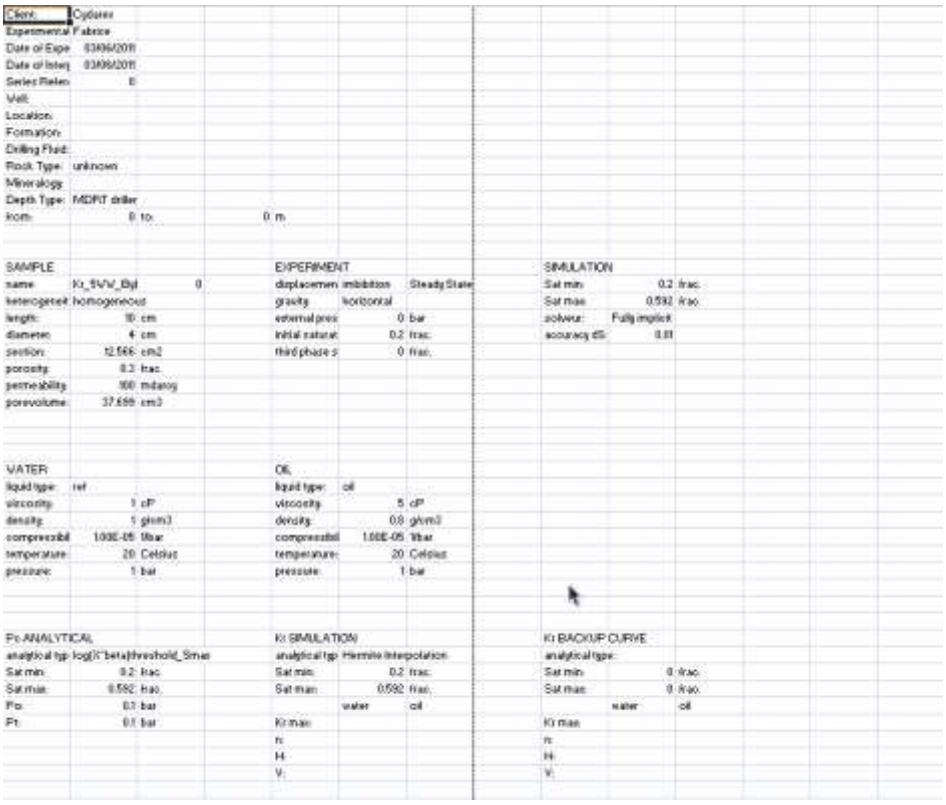


Figure 13: Part of a two-phase flow ASCII report.

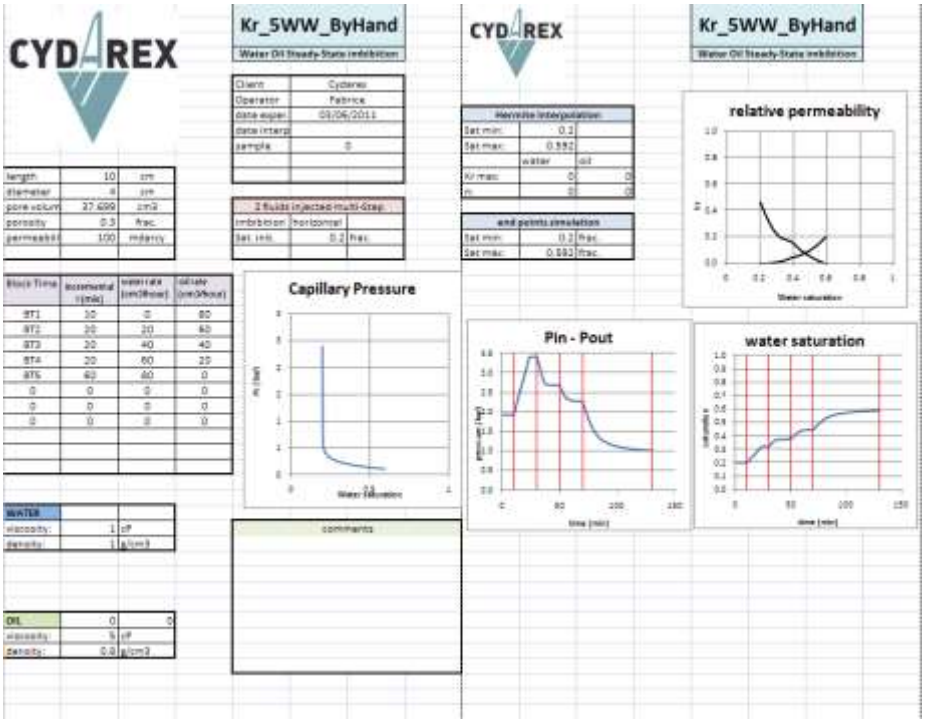


Figure 14: Example of a more complex layout using a template.

Copying a graph or a frame

Select the graph or the frame and press Ctrl+w to copy in the clipboard or use “copy active window” in the edit menu. Then copy in Microsoft Word, Excel, or any other software.

New Project Window

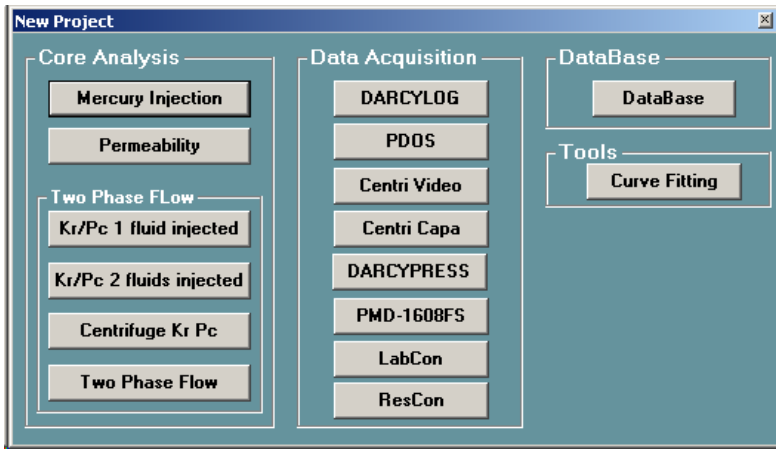


Figure 15: CYDAR new project window.

Mercury injection: interpretation of experiments with calculation of the dS/dP, psd and ds/dLogP curves.

Permeability: absolute permeability calculation for gas and liquid, steady state and transient experiments, using Klinkenberg correction, inertial effect, and correction of pressure drop in tubings.

Two Phase flow frame:

One Fluid Injected: launches the Two Phase Flow module for unsteady state experiments, allowing analytical calculation of the Kr.

Two Fluid Injected: launches the Two Phase Flow module for steady state experiments.

Centrifuge Kr Pc: launches the Two Phase Flow module for centrifugation experiments with calculation of the locale Pc curve from experimental data and the Hagoort Kr.

Two phase flow: Simulation of a large panel of experiments, from gravity flow to unsteady state experiments.

Data Acquisition frame: data acquisition from specific experimental set-up. These modules are optional, and may not be activated on your version.

DataBase: possibility to create a database with different kind of fields, such as rock type, experiment type, client, operator, date. This module is under development and may not be activated on your version.

Curve fitting: access to CYDAR curve fitting tools.

Mercury Injection Module

Mercury injection is used to calculate drainage capillary pressure for initializing reservoir simulations (Leverett J function) and to determine pore size distribution (EOR and rock typing).

Main features:

- direct opening of the ASCII file generated by the experimental apparatus,
- powerful data smoothing using spline functions,
- calculation of 3 types of pore size distributions (PSDs),
- calculation of Pc curves for reservoir fluids and saturations in transition zones,
- calculation of water cut in transition zone when producing the well,
- permeability estimation using Purcell, Swanson[11], and Thomeer[13] methods,
- automated reporting.

Mercury main window

The "**Mercury injection**" button opens the Mercury main window (Figure 16). This window allows loading data, fitting curves, calculating PSD (Pore Size Distribution), and more.

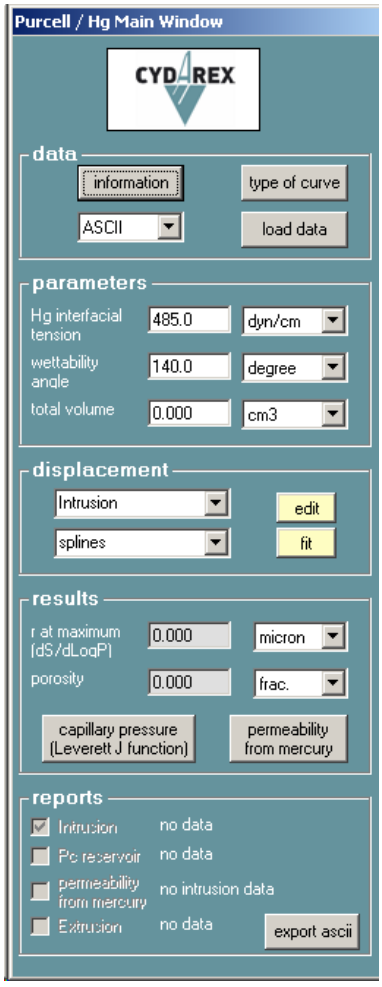


Figure 16: Mercury main window

Data frame:
"**Information**": Common to all modules, used to enter sample name and all information regarding the sample.
"Load data": The list box allows selection of the type of input, either from an ASCII file or from a data file from an apparatus (such as Micromeritics Autopore). Then click on "load data".
"Type of curve": Specifies which parts of the data corresponds to the first intrusion, to the extrusion, and to the second intrusion (detailed below).

Parameters frame:
Allows to enter mercury's interfacial tension and contact angle under the experimental conditions; these vales are used for pore size calculations (Laplace's law). By default the values are given under standard conditions. The total volume (or bulk volume) is optional and used only to calculate porosity.

Displacement Frame:
Fit using linear or splines functions to one of the displacements: intrusion, extrusion, or 2nd intrusion.

Results frame:
"**r at maximum**" gives the value of the "pore radius" from Laplace's law corresponding to the maximum of ds/dLogP.
Porosity is calculated from the volume of mercury injected (pore volume) and the total volume entered in "parameters".

"**Capillary pressure**" button calls the window for calculation of the J Leverett function. It is used to calculate a capillary pressure curve with a different couple of fluids, and the profile of saturation in reservoir transition zones.

"**permeability**" button opens the window for permeability calculation from the mercury Pc curve (Purcell, Swanson[11], …).

Report frame:

- Common to all options: general information, mercury and sample properties, experimental data;
- "intrusion" option: Log(S Hg), Log(P), r, dS/dLog(P), S ref and Leverett function;
- "Pc reservoir" option: reservoir and fluids parameters, Pc reservoir, transition zone, water cut.
- "permeability from mercury" option: Swanson[11] and Thomeer parameters and data.
- "extrusion" option: equivalent to the "intrusion option" but for extrusion data.

Useful tip: Options are enabled only if corresponding data or parameters exist. Options can be partly disabled; in that case, a label indicates which information is missing.

Mercury types of curves

The "type of curve" window (Figure 17) allows the user to choose the type of data. A typical data is composed of 2 columns and n rows. This window also allows the user to remove the beginning of the curve that is not used for interpretation (conformance corrections or surface effects in mercury injection).

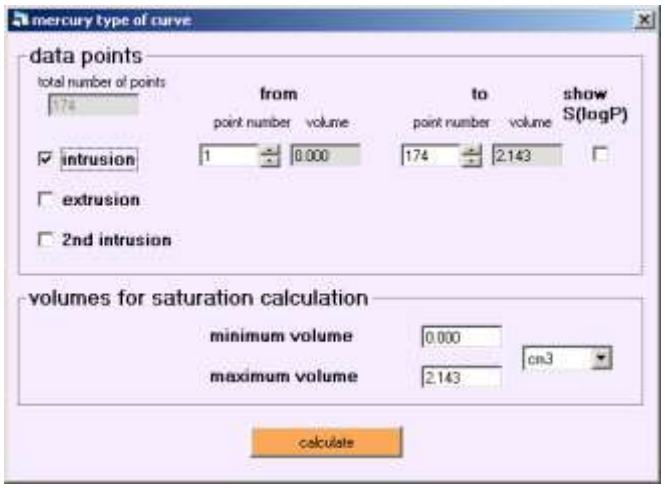


Figure 17: "mercury type of curve" window.

Pore size distribution

The different functions for characterizing the pore size distribution are summarized in Table 2.

pore size distribution $f(r) = \frac{\mathrm{P}^2}{2\gamma\ cos\ \theta}\ \frac{dS}{d\mathrm{P}}$	Characterization of the distribution of the smallest pores, roughness of the walls (fractal dimension). Mainly used to determine the fractal dimension of pore roughness
the logarithmic derivative $dS/d(\ln P)$	Pore topology, mean diameter (to derive permeability), presence of a double porosity. Used in Core Analysis.
the derivative dS/dP	Information for large radius and low capillary pressures.

Table 2: different functions used to characterize the pore size distribution.

CYDAR calculates all functions automatically. Once data are entered, a linear interpolation is performed automatically. The user can then smooth the data curve using a splines fit. From the analytical curve, linear, or splines fit, the three following functions are accessible through the "view" menu (Figure 18): S(LogP), dS/dP, psd and ds/dLogP.

In the view window (right side of Figure 18) the frames called "intrusion", "extrusion" and "2nd intrusion" are enabled if the corresponding types of curve have been selected in the "mercury type of curve" window (Figure 17).

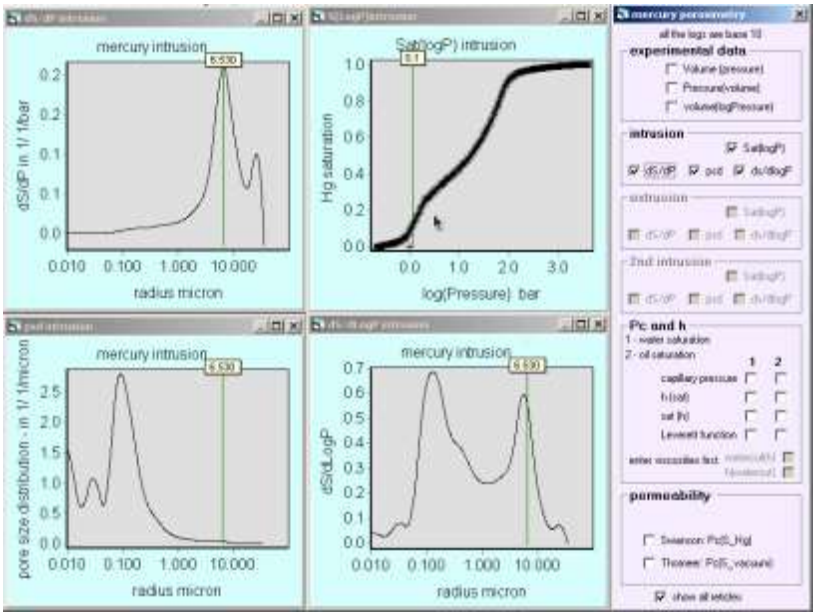


Figure 18: Graph examples in "mercury injection".

Permeability calculation

Permeability can be estimated from the pore size distribution and porosity. The "tube model", Katz and Thompson, Swanson[11] and Thomeer[13] relationships are implemented. The porosity value is necessary for all models.

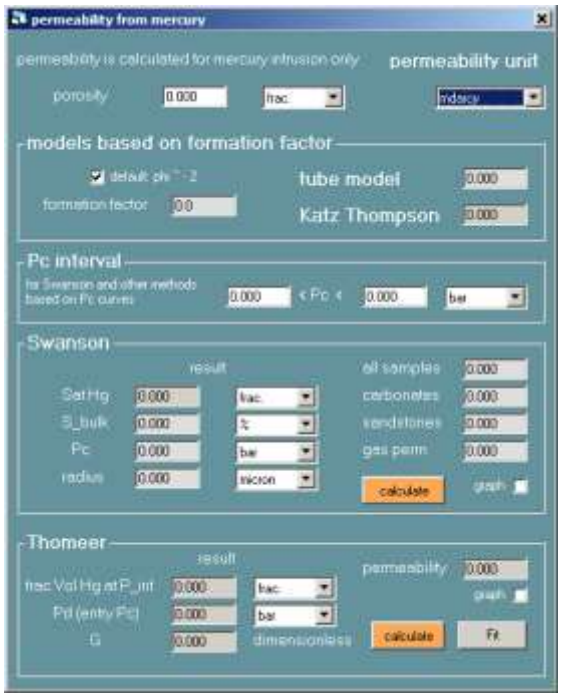


Figure 19: Estimation of permeability from mercury injection.

Models based on formation factor:

Checking "default" gives the formation factor as Φ^{-2}, otherwise the formation factor must be specified. Permeabilities are calculated automatically.

Pc interval:

The range selected is useful in case of double porosity to adjust the interval of pressure for permeability determination.

Swanson:

The 3 brine permeability values correspond to the experimental relationships given in Swanson publication (Swanson, B.F., "A simple correlation between permeabilities and mercury capillary pressures", *Journal of Petroleum Technology, December, 2498-2504, 1981*) for sandstones, carbonates, and an average over all the samples. **"Gas perm"** is the gas permeability relationship shown in Fig. 5 of Swanson's publication.

Checking **"graph"** displays the Swanson graph with Swanson point and tangent (Figure 20).

Thomeer:

The parameters correspond to the original publication. The procedure is the following:

- The "**Fit**" button opens the "modified hyperbolic" window (Table 1d)) for fitting the Log(Pc) vs. Log(SHg) curve. It is recommended to start with the first guess then to optimized parameters a, b and c. The "**Accept**" button keeps parameter values. Alpha is fixed to 1.
- The calculate button determines Thomeer parameters and the permeability.
- Generally the result depends on the Pc interval chosen for the fit of the hyperbola.

The graph checkbox shows a graph with the pressure/saturation curve and the Thomeer equation as analytical fit (Figure 21).

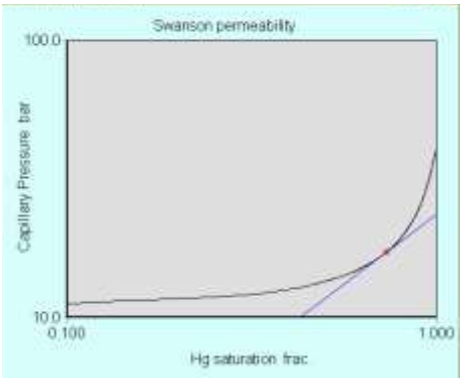


Figure 20: Swanson method: automatic calculation of the tangent.

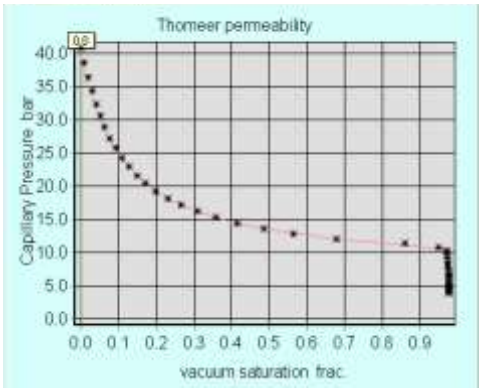


Figure 21: Thomeer method: fit of the Pc(sat) curve by an hyperbolic function in log/log scale.

Capillary pressure from mercury

The "Capillary pressure from mercury" window Figure 22is used to enter parameters for the J Leverett function and Pc curve at reservoir condition, the transition zone, and the water cut. Calculation results can be displayed on graphs using the "view" menu.

J Leverett Function

The Leverett J function is used to calculate a capillary pressure curve with oil and water (or gas) using the mercury capillary pressure (M.C. Leverett. "Capillary behavior in porous solids". *Transactions of the AIME* (142): 159–172, 1941):

$$J = \frac{Pc}{\gamma \cos\theta}\sqrt{\frac{K}{\phi}}.$$

The principle is to assume that the J functions are the same for the two Pc curves. Although the term "reservoir" is used, the calculation can be applied to any experimental conditions. The "reservoir" Pc curve and the Leverett function can be displayed using the "view" menu.

Figure 22: Window for capillary pressure calculation and saturation distribution in reservoir transition zones.

Useful tip: The J Leverett function can be used for capillary pressure curves obtained with other fluids than mercury. The experimental Pc curve is entered as for mercury, and the Hg interfacial tension is replaced by the corresponding value for the couple of fluids. For positive drainage, saturation should be the non-wetting fluid saturation (equivalent to mercury), and the contact angle, the value measured in this fluid (>90°).

Transition zones

Saturation in the transition zones is derived from the balance between gravity and capillary forces:

$$Pc = P_{oil} - P_w = (\rho_w - \rho_{oil})gh.$$

In addition to the parameters used in the J functions, this calculation requires the reservoir oil and water densities.

Water cut

This function is used to estimate the production of the well at a given position in the transition zone. The top of the transition zone produces 100% oil and the bottom only water. The water fractional flow (also called water cut) is derived from Darcy's law:

$$fw = \frac{Qw}{Qw+Qo} = \frac{Kr_w/\mu_w}{Krw/\mu_w+Kro/\mu_o}.$$

Fluid viscosities are entered and relative permeabilities are calculated using the approximation of a Corey function with parameters defined in the window.

Tutorial files

Tutorial_Hg_SampleA

This example uses an ASCII file generated by an AUTOPORE apparatus (Tutorial_Hg_SampleA.rpt) and corresponds to the type of data "Autopore IFP".

In order to calculate the porosity, the user must enter the total volume of 6.5 cc.

The corresponding CYDAR project is "Tutorial_Hg_SampleA.cyd".

Tutorial_Hg_Brauvillier

This example uses an ASCII file from the Microsoft Excel spreadsheet "Tutorial_Hg_Brauvillier.xls".

In order to calculate the porosity, the user must enter the total volume of 7.733 cc.

The corresponding CYDAR project is "Tutorial_Hg_Brauvillier.cyd".

Permeability Module

In most laboratories, absolute permeabilities are generally calculated with a Microsoft Excel spreadsheet using permanent flow experiments. Using CYDAR minimizes the risk of errors, improves the quality control, and allows the permeability determination from transient flow.

CYDAR includes the following options:

- Steady state and transient flow experiments,
- Determination of inertial coefficient (Forchheimer correction),
- Determination of Klinkenberg correction,
- Transient flow experiments, with the possibility to take into account experimental setup dilatation due to pressure,
- Automated reporting to export results in an Excel file.

Permeability main window

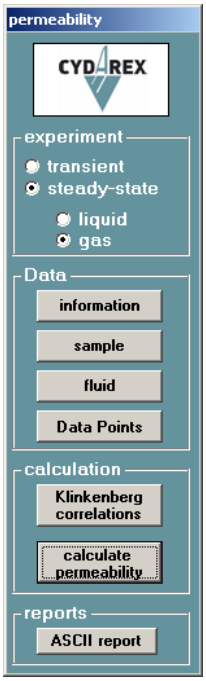


Figure 23: Permeability Main Window.

Experiment frame:
"**Transient**": pulse decay experiments.
"**Steady-state**": permanent flow experiments.
"**Liquid**": constant compressibility (transient).
"**Gas**": the fluid is compressible, following Boyle's law (ideal gas) or real gas.

Data frame:
"**Information**": common to all modules, allows entering information regarding the sample and experience. None of this information is used for calculation.
"**Sample**": dimensions and porosity of a cylindrical sample.
"**Fluid**": fluid properties window.
"**Data points**": input data spreadsheet, see below.

"Klinkenberg correlations" button displays a panel to estimate Klinkenberg coefficient (see section on "Klinkenberg correlations").

"Calculate permeability" determination of permeability with four possible corrections: inertial, Klinkenberg, tubing, and temperature (see sections on "Calculate permeability: steady-state" and "Permeability pulse")

Graphs are displayed using the "**view**" menu.

Sample and fluid properties

The sample is assumed to be cylindrical. In steady-state, the porosity is optional, and is only used in calculating the Reynolds number. In addition to the length, the user needs to enter one of the following: the diameter, the section, or the pore volume if the porosity is known (Figure 24). Because the porosity can be optimized in the transient module, its input is on the calculation window (Figure 23). Figure 25 shows the fluid properties windows.

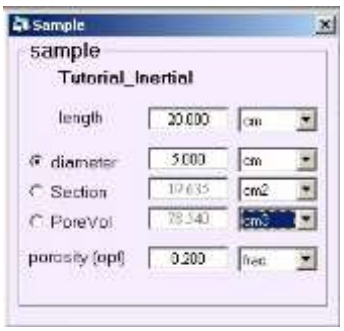


Figure 24: Sample window.

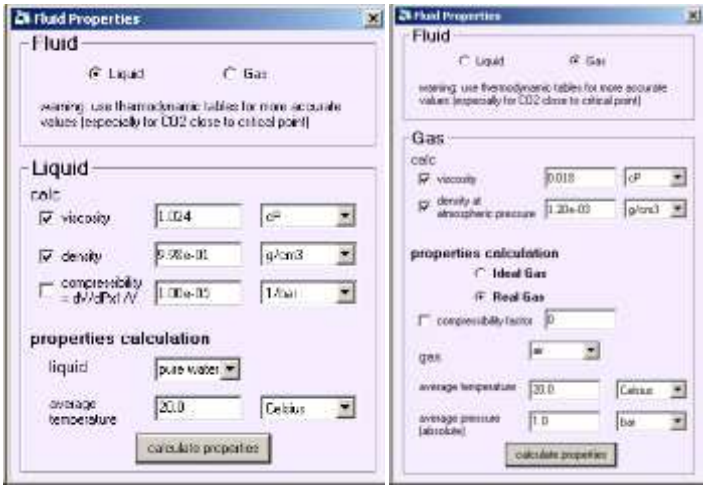


Figure 25: "Fluid properties" windows.

Klinkenberg correlations

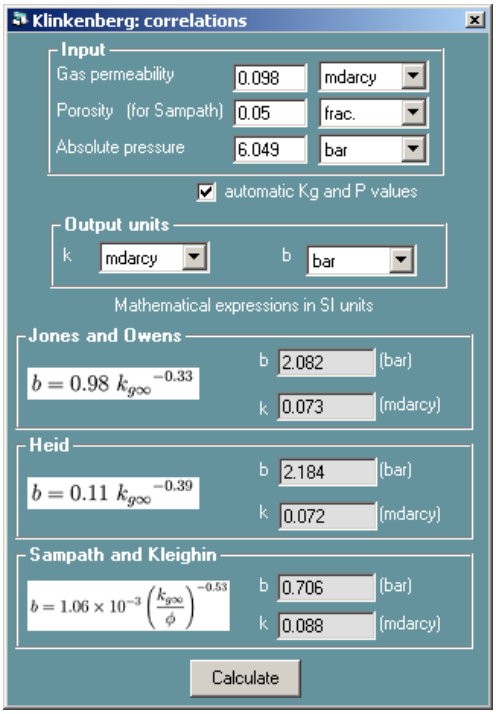


Figure 26: Klinkenberg correlations window.

This window allows the Klinkenberg coefficient calculation from well-known correlations. The values are only output, they are not used in CYDAR calculation.

"**automatic Kg and P values**" option set the gas permeability to the permeability set or calculated, in the "calculate permeability" window (Figure 23) and the absolute pressure.

The automatic absolute pressure is:
In a steady-state experiment, the average between the maximum inlet pressure and the minimum outlet pressure.
In a transient experiment, the average of the averages between maximum and minimum of each data.

The different models used are described in the corresponding journal articles, given in the Reference section below.

For instance, the Jones & Owens[15] correction uses the following equation:

$$b = 0.86\ K_{g\infty}^{-0.33}$$

Steady-state experiment

Data points steady-state

The "data points" button on Figure 23 opens a spreadsheet (Figure 27 and Figure 28). Data can be added by copy/paste from other programs such as Microsoft Excel, CYDAR spreadsheet, a text file, or manually typed. Columns "date", "hour", "volume Q", "symbol" and "comments" are always present.

Pressures:

- Either the two-pressures "**P inlet**" and "**P outlet**" are measured, or the differential pressure "$\mathbf{P_{inlet}}$-$\mathbf{P_{outlet}}$" is entered.
- If the differential pressure is measured, inlet or outlet pressures must be added to the relative pressure drop.
- A pressure can be entered as a "**constant**" (outlet pressure for instance) or "**measured**". In the later case, the corresponding column appears in the table and must be filled.

Flow rate:

- The flow rate can be a mass or a volume flow rate.

Columns:

- "**date**", "**hour**" and "**comments**" are optional.
- "**symbol**" allows entering two sets of data points with symbol 1 or 2 (or 0), that will be displayed with different symbols in the graph, for increasing and decreasing values for instance.

For a gas, the pressure Pq is the pressure at which the flow rate is measured.

IMPORTANT: All pressure values are "relative pressures" (relative to atmospheric pressure). P=0 when at atmospheric pressure. Only "atmospheric pressure" is an absolute pressure.

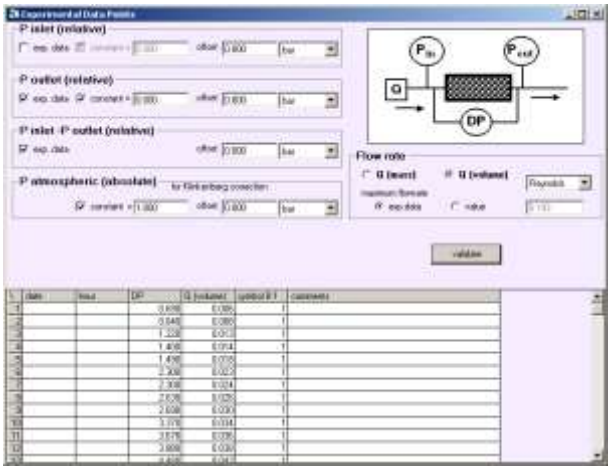


Figure 27: Data points spreadsheet for a liquid.

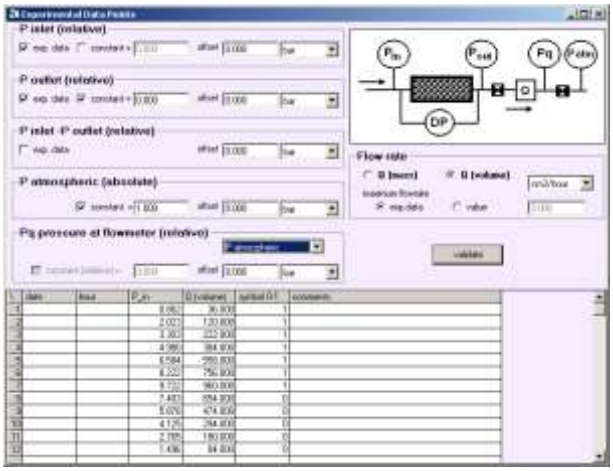


Figure 28: Data points spreadsheet for a gas.

Calculate permeability: steady-state

For a liquid, Darcy law gives:

$$\frac{Q}{A} = -\frac{K}{\mu}\left(\frac{(P_{out}-P_{in})}{L} + \rho g \cos\theta\right).$$

The variable Y (Figure 29) is defined as:

$$Y = P_{in} - P_{out}$$

For a vertical flow with injection at the bottom ($\cos\theta = 1$), at equilibrium (Q=0), the above Darcy's law is reduced to the hydrostatic law:

$$P_{in} = P_{out} + \rho g L.$$

Generally, the gravity term, measured at equilibrium, is subtracted from the inlet pressure (offset pressure).

For a gas, gravity is always negligible:

$$\frac{Q}{A} = -\frac{K}{\mu}\frac{1}{2P_q}\frac{\left({P_{out}}^2-{P_{in}}^2\right)}{L},$$

where Q is the flow rate measured at the pressure Pq. The variable Y is defined as:

$$Y = \frac{P_{in}{}^2 - P_{out}{}^2}{2P_q}.$$

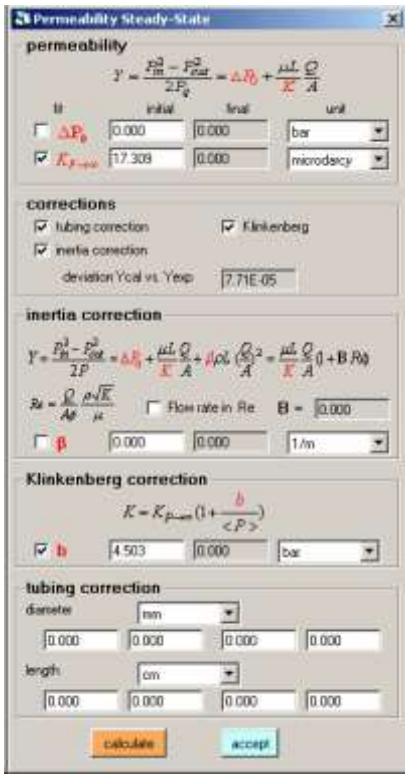


Figure 29: Permeability calculation.

The "**calculate permeability**" button opens Figure 29.

The section "**permeability**" allows fitting the data by adjusting the permeability and/or the offset pressure.

The check boxes in the "**Corrections**" frame allow other frames to appear: Inertia, Klinkenberg, and tubing corrections.

Parameters with checked boxes are adjusted when clicking on the "**calculate**" button.

If the "**fit**" box is not checked, the calculation is performed with the initial value (non zero value for permeability).

The "**view**" menu displays various graphs, with both experimental and calculated data.

For gases or liquids with inertial effects, the function Y cannot be calculated as a continuous curve if one of the pressures (inlet or outlet) is not constant.

In all cases, the agreement between the experimental and calculated data is displayed in the graph "**Y_{calc} vs. Y_{exp}**".

The physical meaning of using both inertial and Klinkenberg corrections is not validated.

Inertial correction

For high flow rate, inertial effects reduce the section of passage of the fluid through the pores, and the "apparent" permeability decreases as the velocity increases. This effect is more pronounced for high permeability (K > 100 mDarcy) and gas. The most popular empirical law to model experimental data was established by Forchheimer as follow (Forchheimer 1901):

$$\frac{(P_{in}-P_{out})}{L} = \frac{\mu U}{K} + \beta\rho U^2,$$

where β is called the inertial coefficient and has the dimension of the inverse of a length. This equation is valid for an incompressible fluid. For a gas, the right term must be replaced by $\left(P_{in}{}^2 - P_{out}{}^2\right)/\left(2P_qL\right)$. Forchheimer law assumes that the pressure gradient depends on a viscosity term (Darcy's law) and on an inertial term in U^2.

The inertial coefficient β can be determined by optimization together with K by checking the corresponding boxes.

These equations can also be written in dimensionless form using the Reynolds number:

$$\mathrm{Re} = \frac{\rho V d}{\mu},$$

where d is a characteristic length defined by $d = \sqrt{K_d}$ in porous media, K_d is the permeability at low velocity, and V is the interstitial velocity defined as $V = U/\phi = Q/(\phi A)$. Therefore:

$$\mathrm{Re} = \frac{\rho Q\sqrt{K_d}}{\mu\phi A}.$$

By introducing a dimensionless coefficient define as $B = \beta\phi\sqrt{K_d}$, we obtain:

$$\frac{(P_{in}-P_{out})}{L} = \frac{\mu U}{K}(1 + B\mathrm{Re}).$$

Typically the inertial effect becomes significant when $\mathrm{Re} \geq 1$. On the graph, the flow rate can be displayed in the unit of Reynolds numbers, to verify if the inertial correction is justified.

Klinkenberg correction

The principle of the calculation is a non-linear optimization on all points to determine simultaneously the "liquid" permeability K_L that corresponds to the limit of infinite pressure, and the Klinkenberg parameter *b*. It is recommended to set the offset ΔP_0 to zero for the Klinkenberg calculation:

$$\frac{Q}{A} = -\frac{K_G}{\mu}\frac{1}{2P_q}\frac{(P_{out}{}^2 - P_{in}{}^2)}{L},$$

with the gas permeability defined as:

$$K_G = \mathrm{K_L}\left(1 + \frac{\mathrm{b}}{\langle P \rangle}\right).$$

The optimization is performed using a Levenberg-Marquardt method from the IMSL Fortran Library. Due to the method of calculation, all the calculated points are always exactly on the line *K*(1/<*P*>) on the Klinkenberg graph. The graph "**Y_{calc} vs. Y_{exp}**" is used to control the fit accuracy.

Tubing correction

The "**tubing correction**" option calculates the pressure drop in the tubing using Poiseuille's law, and subtracts this pressure to the measured pressure. This effect is noticeable with tubings of small diameters. The window allows entering four different tubing with diameter and length, mounted in series.

Transient experiment

Data points transient / unsteady state

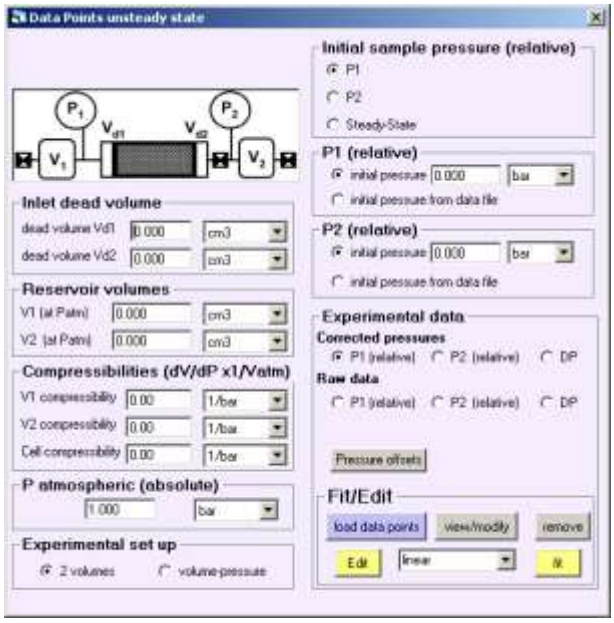


Figure 30: Data points transient window.

IMPORTANT: All pressures are relative to the atmospheric pressure, except the atmospheric pressure itself, which is absolute.

Experiment types:

- Two volumes: input and output are controlled by tanks with known volumes and given initial pressures.
- One volume and one pressure: one side is controlled by a tank with known volume and a given initial pressure, the other side by a pressure.
- In each case, the side numbered 1 is always a volume. Only the side numbered 2 can be either a volume or a pressure.

Initial sample pressure:

- Uniform pressure: the sample is at one of the initial pressures P_1 or P_2.
- Non uniform pressure: the initial pressure profile corresponds to a steady-state flow. The profile calculation requires fluid and sample properties, as well as the flow properties like Klinkenberg coefficient and permeability entered in the calculate window, see section Permeability pulse.

P_1 and P_2 pressures:

- P_1 is always an initial pressure either entered by the user or read from data.
- With a two volumes experiment, P_2 is, as P_1, an initial pressure given by the user or read from data. Otherwise, P_2 is a pressure versus time either constant or read from data.

Experimental data:

- Experimental data are relative pressures versus time: P_1, P_2 or $DP = P_1-P_2$.
- Raw data: Raw data are experimental data. They cannot be edited or modified. From those data, a few corrections can be done such as offset and averaging. By clicking on the "pressure offsets" button the corresponding window is opened. Resulting calculation gives the corrected pressures.
- Corrected pressures: can be directly loaded or calculated from raw data. These pressures can be edited and fitted (see section on Data editing). These data are used for simulation and optimization.

Compressibilities: "**Compressibilities**" allows taking into account the experimental setup dilatation due to pressure differences with the atmospheric pressure. The volume variation is assumed to be linear and the compressibility χ is defined as follow:

$$\frac{(V-V_{atm})}{V_{atm}} = \chi(P - P_{atm}).$$

Permeability pulse decay

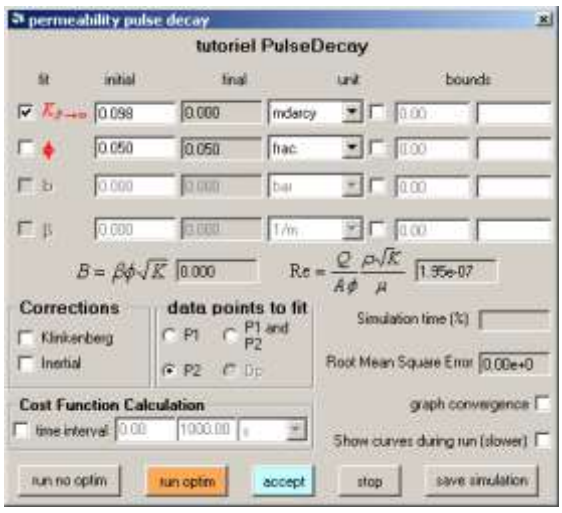


Figure 31: Calculation panel for transient permeability.

The mass balance equation is solved in one dimension. Klinkenberg correction and inertial effects can be used:

$$\phi c \frac{\partial P}{\partial t} + \frac{\partial \phi}{\partial t} + Uc \frac{\partial P}{\partial x} + \frac{\partial U}{\partial x},$$

$$-\frac{\partial P}{\partial x} = \frac{\mu}{K} U + \beta \rho U^2,$$

$$K = K_L\left(1 + \frac{b}{\langle P \rangle}\right).$$

The numerical scheme is implicit and the system is solved by a Newton-Raphson algorithm. The cost function is calculated with P_1, P_2, both, or $DP=P_1-P_2$ (frame "data points to fit" in Figure 31).

Buttons:

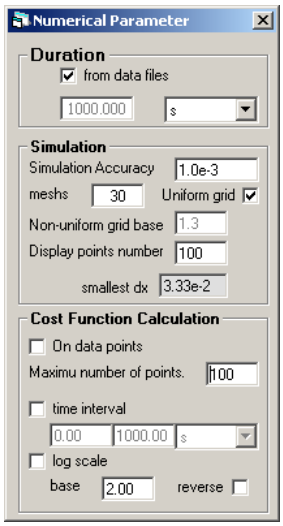
- "**run no optim**": runs one simulation without any optimization.
- "**run optim**": runs optimization on selected parameters: permeability, porosity, Klinkenberg coefficient and/or inertial coefficient.
- "**accept**": accepts optimized values. This action erases the first guess values.
- "**stop**": stops simulation and optimization.
- "**Back up curve**": back up the current simulation curves.

The user can give bounds to each parameter using the corresponding check boxes.

Useful Tips: Although permeability, porosity, Klinkenberg, and inertial coefficients can be optimized all together, it is recommended to perform optimization one parameter at a time.

Parameter menu transient:

In transient type experiment, an additional menu is enabled to access numerical parameters (Figure 32):



"**Duration**": duration of the simulation.
"**Simulation**": simulation accuracy on pressure (adimensional).
"**Mesh**": grid number.
"**Uniform grid**": if not selected, the mesh cells grow progressively from the outlets to the sample center symmetrically as follow: $dx_i = r dx_{i-1}$. The user gives *r* as the "non-uniform grid base".
Display points number: sets the number of points of the simulation curve displayed on graphs.

The "**smallest dx**" is a read-only value giving the smallest dx existing in the mesh.

"Cost function calculation": By default, the cost function is calculated over the entire time interval defined above on a linear time scale. However, the user can define a time interval and a time log scale.

Figure 32: Numerical parameter window for absolute permeability calculation in transient flow.

Tutorial files

Tutorial_Perm_Klinkenberg

Data of the steady-state Gas tutorial are in the Microsoft Excel file "Tutorial_Perm_Klinkenberg.xls". The corresponding CYDAR project is "Tutorial_Perm_Klinkenberg.cyd"

Tutorial_Perm_Inertial

Data of the steady-state liquid tutorial are in the Microsoft Excel file "Tutorial_Perm_Inertiual.xls". The corresponding CYDAR project is "Tutorial_Perm_Inertial.cyd".

Tutorial_PERM_Pulse_decay_Analytical

Data of the transient gas tutorial are in the Microsoft Excel file "Tutorial_PERM_Pulse_decay_Analytical.xls".

The corresponding CYDAR project is "Tutorial_PERM_Pulse_decay_Analytical.cyd".

Two-phase Flow Module

Immiscible flows (two-phase flow, noted TPF) are the most important part of core analysis. The two-phase flow modules of CYDAR allow two complementary approaches:

Direct simulation: To simulate an experiment before running it, to determine the optimal experimental parameters (duration, injection flow rate or pressure). In this approach, the parameters such as relative permeabilities (Kr) and capillary pressure (Pc) are assumed to be known.

Parameter estimation: To use experimental data to determine unknown parameters, mainly Kr and Pc. This can be done by using analytical approaches, numerical optimization, or history matching.

The two-phase flow functions are described in the following sections of this manual and organized as follows:

- **TPF direct simulation**: how to enter the parameters to simulate any kind of displacement, injection of one or two fluids, porous plate, centrifuge, etc.
- **TPF optimization**: description of the principle of the optimization used to determine one or several parameters from an experiment

In addition, CYDAR allows specific calculations for standard procedures to determine Kr and Pc:

- **Unsteady-state** experiments for Kr determination. One fluid is injected and the transient productions and pressures are used to determine Kr. Historically, only one step was used and interpreted with analytical equations assuming that capillary pressure was negligible. This method is available and called JBN (and Jones and Roszelle). However, a more accurate method is now preferred, based on a multi-step displacement and using numerical optimization. This method can also lead to the determination of the capillary pressure curve during the same experiment.
- **Steady-state** experiments. This method is based on the simultaneous injection of two fluids at various ratios. As a first stage, an analytical calculation gives an estimate of the Kr curves. Then an optimization with the experimental data is possible to improve result.
- **Centrifugation.** Several results are available from centrifuge experiments. The first is the determination of the real or "local" capillary pressure curve from the production data, using Forbes and spline methods. The centrifuge also allows determination of the Kr, either in a single step (single speed) or multispeed experiments, using numerical optimization
- **Porous plate.** CYDAR is mainly used with simulation to optimize the duration of the various steps of pressures.

Two-phase flow: direct simulations

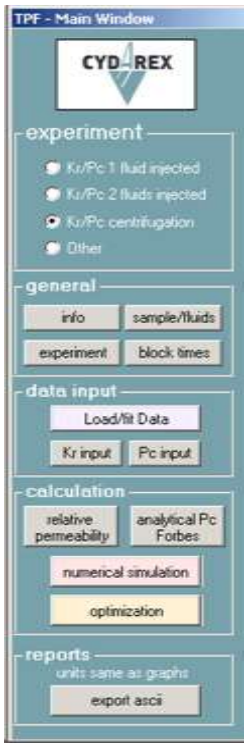


Figure 33: two-Phase Flow (TPF) main window

Experiments frame: The three first options allow the selection of specific experimental parameters in the "**Type of experiment**" window and specific buttons in the "**calculation**" frame. The "**Other**" option means that the experiment type must be set in the "**Type of experiment**" window.

General frame:

"**info**" opens the information window, common to all modules (sample, operator, client …).

"**sample/fluids**" opens the sample and fluid properties window (page 41).

"**experiment**" opens the experiment type settings window (page 48) allowing various type of condition: centrifuge, one or two fluids injection, spontaneous, etc.

"**block times**" (page 49) are simulation parameters depending on the experiment type as: duration time, rotation speed, flow rate.

Data input frame:

"**Pc input**" opens the capillary pressure panel with fit and analytical models (page 41).

"**Kr input**" opens the relative permeabilities panel with fit, Corey model, LET, JBN calculation, etc (page 43).

Calculation frame: The calculation frame gives access to the simulation and optimization windows and specific analytical calculation depending on the experiment type (JBN, steady-state Kr, Forbes, Hagoort).

"export ascii" opens a dialog window to export project as a tab separated ASCII file, which can be open in Microsoft Excel.

Fluids and sample properties

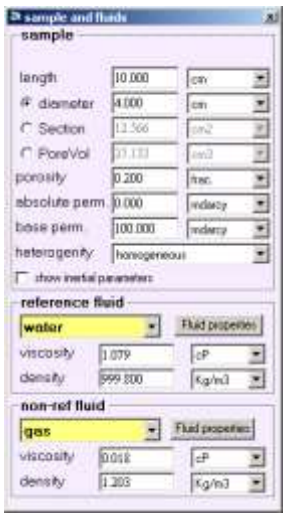


Figure 34: Two-phase flow's "sample and fluids" properties window.

Sample properties:

Common sample properties are geometrical, porosity, and absolute permeability. The base permeability is the one used in simulation (see below).

There are two kinds of heterogeneity:

- Composite means a sample made of several homogeneous blocks.
- Profiles means permeability and porosity are continuous profiles vs. length. The profiles are defined in the "load/fit data" window.

See the "Heterogeneities" section (page 76) for details about data type and how to use heterogeneous sample in simulation.

"Show inertial parameters": the checkbox allows to display in this window, just below, the entries for the inertial parameters (see the "Inertial" page 81 for detail).

Fluids properties:

The "Fluid properties" buttons give access to the fluid properties window described previously (Figure 25).

Definition of the reference fluid:

- Water is always the reference fluid;

Liquid is the reference fluid when the other fluid is a gas;

Base and absolute permeability:

The absolute permeability is the intrinsic permeability. It is not used in calculations. This permeability can be set to zero or left empty.

The base permeability is the one used in simulation and calculation. It is a necessary parameter for simulation. It can be equal to the absolute permeability, but usually, it is a reference permeability which is the value at initial saturation.

For instance, in imbibitions, the base permeability could be the oil permeability at Swi leading to a maximum relative permeability for oil of 1. If the absolute permeability is 100 mD with $Kr_o(Swi) = 0.9$, then the effective permeability is 90 mD . Which is equivalent to a base permeability of 90 mD and Kro = 1. The product (90 mD) is also equal to the effective permeability.

Capillary pressure: Pc

This panel provides different options to implement the capillary pressure curves analytically or from existing data (Figure 35).

If the selected Pc curve in "**Pc type**" has a LogBeta fit, or is an analytical function LogBeta, then a side panel with the LogBeta's parameters is displayed (Figure 35).

Pc type

- The local Pc is the "classical" capillary curve used in simulation when no hysteresis is taken into account.
- hysteresis: the imbibition and drainage curves used to generate intermediate Pc curves (see section "
- Hysteresis")

"Data points" frame

- A label displays the number of data points, if data points have been loaded (see section "**Load/fit data**") or calculated (see section "**Local Pc from average**").
- "**Load data points**" is identical to the button "**Load/fit data**" (see section "Load/fit data").
- "**fit data points**": see section "**Load/fit data**".
- From centrifugation: open the "**Pc Centri**" window (Figure 60).

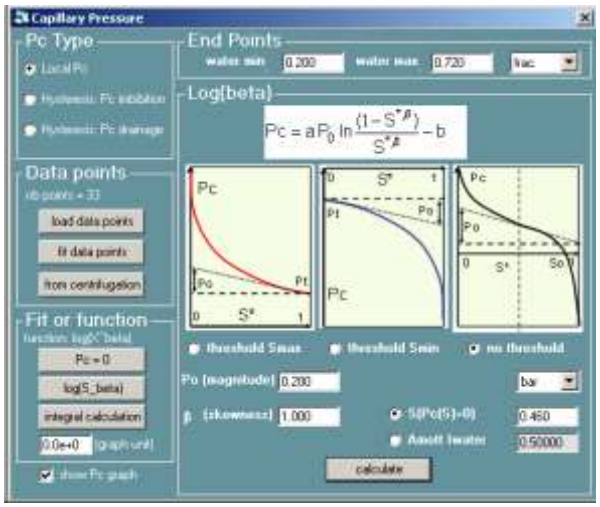


Figure 35: Capillary pressure window, with its side panel when Log(S_beta) has been checked.

Analytical function

For parameter estimation, an analytical function is chosen with two properties:

- A limited number of parameters,
- A physical meaning for these parameters.

This function, called log(beta), is defined as follows:

First, the rescaled saturation is defined:

$$S^* = \frac{S - S_{\min}}{S_{\max} - S_{\min}}.$$

The general function with no threshold is defined by:

$$Pc = -\frac{A}{\beta} Po \left[\ln \frac{S^{*\beta}}{1 - S^{*\beta}} - \ln \frac{(1-S^*)^\beta}{1-(1-S^*)^\beta} \right] + b .$$

As "physical parameter", the magnitude Po and the saturation S_0 are used which correspond to $Pc = 0$, related to b by:

$$b = \frac{A}{\beta} Po \left[\ln \frac{S_o^{*\beta}}{1 - S_o^{*\beta}} - \ln \frac{(1-S_o^*)^\beta}{1-(1-S_o^*)^\beta} \right] .$$

The parameter A is calculated for continuity with the limit cases with threshold defined below.

Threshold at the maximum value of saturation ($S^* = 1$), either in drainage or imbibition:

$$Pc \le Pt \rightarrow S^* = 1 \quad ; \quad Pc > Pt \rightarrow Pc = Po \frac{1}{S^*}.$$

At minimum of saturation ($S^* = 0$):

$$Pc \ge Pt \rightarrow S^* = 0 \quad ; \quad Pc < Pt \rightarrow Pc = -Po \frac{1}{(1-S^*)}.$$

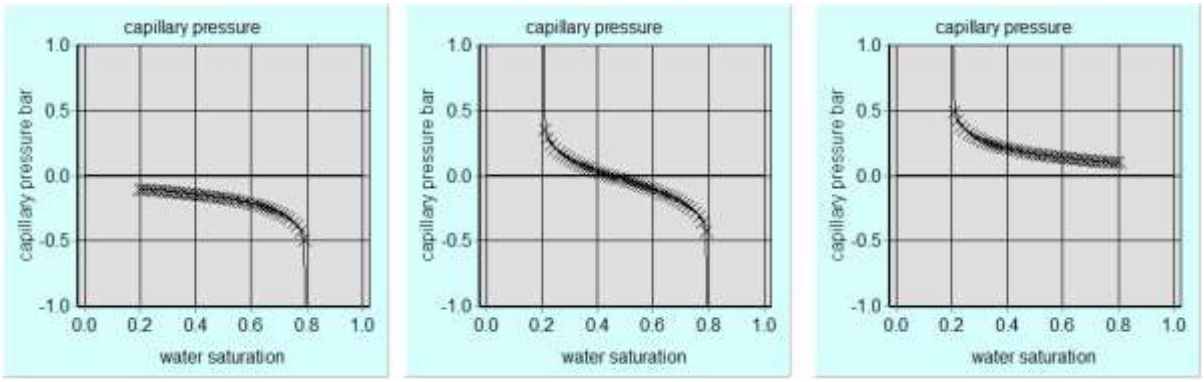
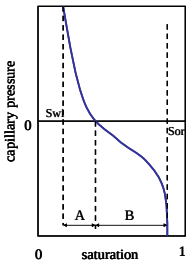


Figure 36: Examples of Pc curves obtained with the analytical Log(beta) function with 3 options: threshold S_{min} (Pt=100 mbar); no threshold (Sat=0.45 at Pc=0), and threshold S_{max} (Pt=100 mbar). For the three curves the magnitude is P_0=100 mbar.

Amott Index

Instead of fixing the point at which the Pc curve crosses Pc(S) = 0, the Amott index can be entered (Amott, E., "Observations relating to the wettability of porous rock", Trans. AIME 219, pp. 156–162, 1959.).

For an imbibition, the Amott index is defined as I_w = A / (A+B). For a drainage, the Amott index is defined as I_o = B / (A + B).



Relative permeabilities: Kr

The relative permeabilities control panel (Figure 37) gives access to several ways for entering Kr curves:

- Experimental Data Points: identical to the button "load/fit data".
- Corey and Modified Corey: Power law models.
- LET: the LET function, as described in "A new versatile relative permeability correlation", Lomeland et al., SCA, 2005 (see Reference 4).
- JBN: analytical calculation with unsteady state experiment.
- Kr fit: allows entering numerical points with fitting via "load/fit data".

End Points

- Maximum relative permeabilities.
- Interval of saturation.

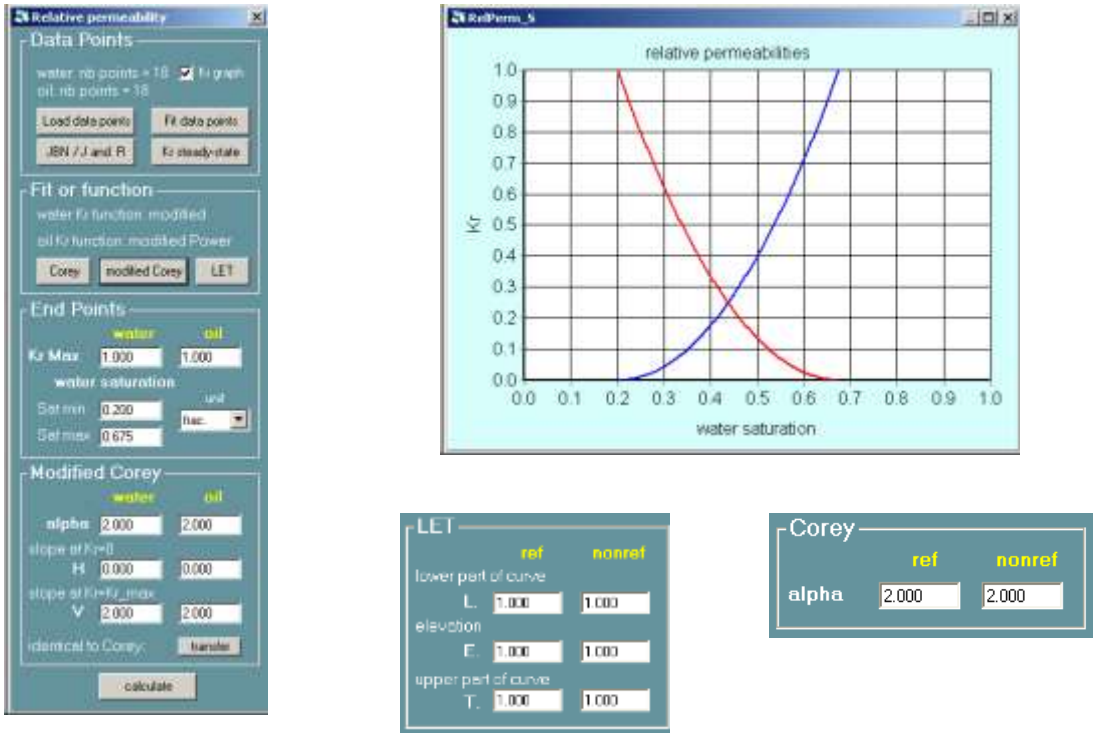


Figure 37: main window for relative permeabilities. Corey option, modified Corey, and LET function with the corresponding kr curves.

Corey model

The entries are the parameters of the Corey's model:

$$S^* = \frac{S_r - S_{\min}}{S_{\max} - S_{\min}} \quad ; \quad Kr_r = K_{r,\max}(S^*)^{\alpha_r} \quad ; \quad Kr_n = K_{n,\max}(1 - S^*)^{\alpha_n}.$$

Modified Corey

The modified Corey function allows the slopes of the kr curves to be adjusted at the two extreme saturations using the parameters *H* and *V*. See example in Figure 37.

The transfer button generates a Modified Corey curve identical to the Corey curve. This option is useful if an optimization using a Corey curve was performed, and the user wants to perform an optimization using a Modified Corey. The starting point for the Modified Corey is now the optimized Corey function.

LET Function

The LET function is described in "A new versatile relative permeability correlation", Lomeland et al., SCA, 2005 (Reference 4), and is usually used for relative permeabilities.

Load/fit data Window

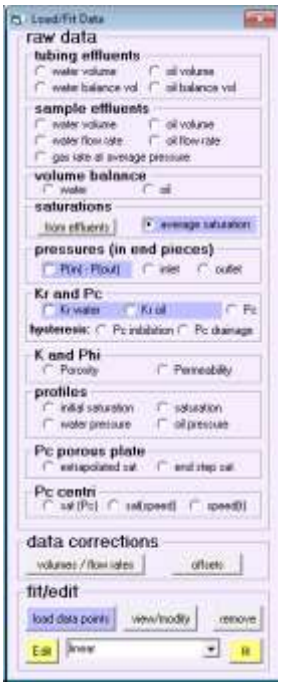


Figure 38: TPF Load/Fit data points window.

Raw data frame: This allows the user to select data to be loaded or edited.

> **Useful Tips:** When values exist, the name is displayed in a darker background. The saturation is always the reference fluid saturation: water with oil, or liquid in a gas/liquid system.

Data corrections frame:

The user can perform several data corrections, such as outlet dead volume or offsets. During such corrections, all data points are modified.

Fit/Edit frame: Once a data is selected:

"**Load data points**" opens a new spreadsheet.
"**View/Modify**" opens the existing spreadsheet.
"**Remove**" erases the data.
"**Edit points**" edits the data, see the data editing section (Data editing): interval, removing, adding or moving points, etc.

"**Fit**": fit type specification.

Raw data

Tubing effluents:

- Production volumes at the end of tubings. Tubing corrections can be made by entering total tubing volumes in the Experiment windows.

Sample Effluents data:

- Volume and flow rate productions at end sample versus time of both fluids.
- Gas rate at average pressure is gas production flow rate versus time calculated at the mean value of the two outlet pressures.

> **Useful Tips:** If the numerical solver is set to uncompressible, the simulated gas rate at average pressure is simply the gas rate at the outlet pressure. To get the real gas rate at average pressure, the numerical solver must be set on compressible.

Volume balance data:

- Net volume balance between inflow and outflow versus time for both fluids.

Saturation data:

- Average saturation of the reference fluid versus time. Water is always the reference fluid and gas is always the non-reference fluid.
- The button "from effluents" allows calculation of the average saturation from production data depending on the boundary condition (Table 3).

The calculation cannot be performed with the gas rate at average pressure.

Experiment type	Inlet boundary conditions	Mandatory data
2 fluids injected: constant flow rates 1 fluid injected: constant flow rate Spontaneous Semi-dynamic (SDM)	2 flow rates (or 0)	One of the production data: volume, flow rate or volume balance.
2 fluids injected: flow rate, pressure. 1 fluid injected: constant pressure Gravity Centrifugation Porous plate	flow rate (or 0) - pressure	One of the production data of the injected fluid at constant flow rate: volume, flow rate or volume balance.

Table 3 Necessary data to calculate the average saturation according to the experience type.

Pressures data (end-pieces):

- Inlet and outlet pressures in end-pieces and their difference versus time.

If outlet pressure data exists and is not set to "constant pressure" in the "type of experiment" window, the analytical fit of the data is used as simulation input.

Kr and Pc data:

- Relative permeabilities and capillary pressure data versus reference fluid saturation.
- Hysteresis: the imbibition and drainage curves used to generate intermediate Pc curves (see section "Hysteresis")

K and Phi data: Porosity and permeability data versus length.

Two types of data can be loaded as composite and profile. The composite type is versus length of blocks. The profile data is versus length.

In the composite case, raw data are used as input in simulation; therefore no editing is possible. The sample length is calculated from the sum of lengths.

In the profile case, common editing is possible. If there is no value at 0 or at the sample length, they are linearly extrapolated. See section "Heterogeneities".

Profiles data:

Initial saturation versus length can be use as input for simulation (see section "

- Hysteresis").
- Saturation and pressure profiles versus length. These data cannot be edited; they can be use for optimization.

Pc porous plate:

- "Extrapolated sat" is the capillary pressure at the extrapolated saturation value calculated from fit (Multi-Step bi-exponential fit);
- "end step sat" is the capillary pressure at the experimental end of step saturation value.

See section Porous Plate p. 75 for details.

Pc centri:

- Sat(Pc) is the average saturation versus capillary pressure;
- Sat(speed) is the average saturation versus rotation speed;
- Speed(t) is the rotation speed versus time. It can be used as simulation input instead of block time values.
- Capillary pressure is calculated from rotation speed, therefore loading Sat(speed) will automatically calculate Sat(Pc).

Data depending on Saturation

For Kr, Pc, Saturation profiles, and average saturation, the saturation can be entered as the reference saturation, the non-reference saturation, or the liquid saturation (Figure 1). The liquid saturation is defined as the sum of the reference saturation and the saturation of the third immobile phase:

$$S_{ref} + S_{nonref} + S_{immo} = S_{liquid} + S_{nonref} = 1.$$

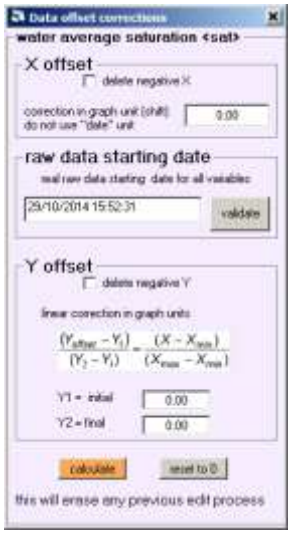
When loading data, S_{immo} can either be a constant value defined in the "**Type of Experiment**" window (Figure 39), or a variable data entered in an additional column.

> **Useful Tip:** Set the value of the third phase S_{immo} (in "**Type of Experiment**" window) before entering saturation data.

If value of S_{immo} is modified after data have been entered, please double check that saturation data have been modified accordingly. CYDAR usually recalculate the non-reference saturation and keeps the reference saturation unchanged.

Data corrections

Offsets: The offsets calculation must be performed first, before any correction or editing. Any action on this window ("calculate" or "reset to 0") will erase all previous changes.



- "**X offset**" is the abscissa offset. All data will be shifted by X_{offset}.
- "**Y offset**" is the ordinate linear offset. It corresponds to a linear de-trending of the y-coordinate by a line passing by Y_1 and Y_2.
- "**delete negative X or Y**" deletes any negative value after calculation.
- "**Reset to 0**" button: reload the raw data.

When the x-variable is time, the starting time of the raw data is displayed and can be modified in the corresponding text box (press validate and calculate)

The beginning of the experiment can be changed using the X offset. For instance, setting the x-offset to -60 s, will start the experiment 60 seconds before the first recorded point (useful for centrifuge when the first data point is recorded when the speed is stabilized).

However, the raw, corrected and edited data keep the same "date". For the edit graphs (yellow background), the date format corresponds only for the edited points. For the XY display of the edited points, the date format is not refreshed automatically. When the starting date, or time offset have been changed, the time axis must be changed to seconds or minutes, then to "date" again to refresh the calculation of the date.

Correction volumes and flow rates

Dead volume correction on volumes and flow rates:

- One outlet tubing;
- One fluid initially.

The correction must be performed after offset calculation and before any editing. The calculation will erase all changes but the offsets.

Type of experiment

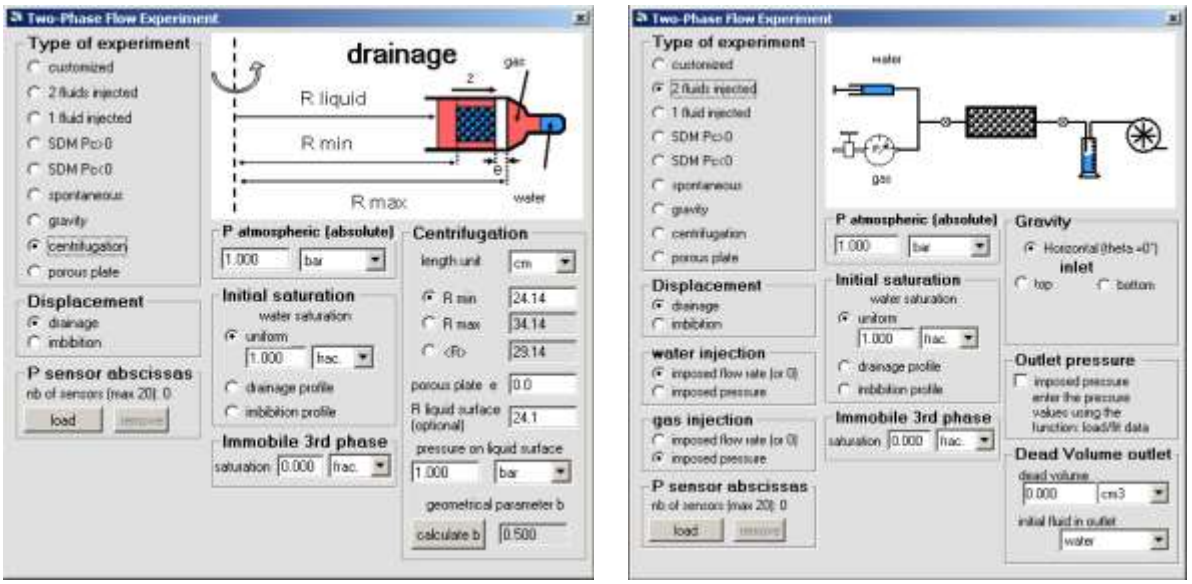


Figure 39: Type of experiment windows: a centrifuge case (left) and a steady-state case (right).

Different experiment types can be selected. Unless using the customized version, the boundary conditions are automatically determined by the software. Note that the initial reference saturation must be between the two end points values.

"**P atmospheric (absolute)**" is the only pressure in absolute value; all other values are relative to atmospheric pressure.

"**Initial saturation**" is the initial water saturation. If it is non-uniform, the data loaded in the data point window are used. A non-uniform initial saturation can be used in all experiment cases, however its main purpose is to be used with the hysteresis option (see section "Hysteresis"):

- "**uniform**": the sample is at the value entered here in all grids.
- "**drainage profile**": the initial profile results from a drainage experiment.
- "**imbibition profile**": the initial profile results from an imbibition experiment.

"Immobile 3rd phase" is used to take into account the presence of an inert, immobile third phase in the sample. By its presence in the sample, the 3rd phase reduces the saturation. For instance, a 10% 3rd phase means that the two other fluids can only occupy 90% of the sample. The usual case would be oil/gas in presence of a third immobile phase composed of water.

Useful Tip: Set the value of the third phase S_{immo} (in "**Type of Experiment**" window) before entering saturation data. See section "**Data depending on Saturation**" above.

"Gravity" is used to take into account gravity effects, depending on the orientation of the sample.

"Outlet Pressure" fixes the pressure at the outlet of the sample. It is a relative pressure; therefore zero outlet pressure corresponds to atmospheric pressure.

"Dead Volume Outlet" is used to take into account dead volumes introduced by tubing at the outlet of the sample. The total dead volume and the initial fluid that it contains must be entered. Then volumes and volume balances at the exit of the tubing can be calculated.

Block times

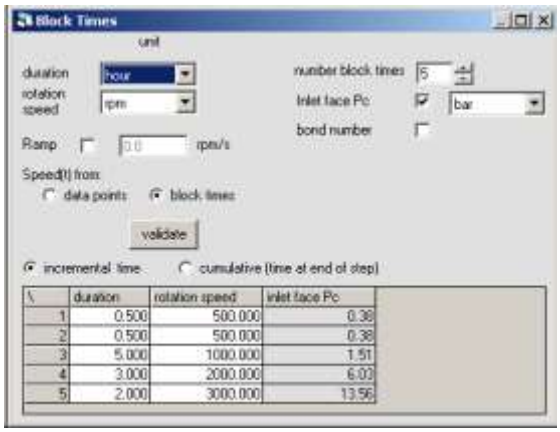


Figure 40: Block times window, centrifuge case.

The "**block times**" window is an input panel for simulation. It defines the different time steps. The user can add or remove lines using the edit menu or the right click menu, or by using the "**number of block time**" function.

The different entries, depending on the experiment type, are summarized in Table 4. The column "n" is the block time number. With the customized option, entries depend on the inlet boundary conditions selected.

In the centrifuge case, there is also the possibility to use experimental data input as rotation speed versus time and a ramp at beginning of each block times.

Experiments	Entries
Spontaneous	n: 1 — duration: 0.000
Porous plate	n: 1 — duration: 0.000 — Ref Fluid pressure: 0.000 — Nonref Fluid press: 0.000
Kr Unsteady State	n: 1 — duration: 0.000 — Ref Fluid pressure: 0.000
Kr Steady State	n: 1 — duration: 0.000 — Ref Fluid rate: 0.000 — Nonref Fluid rate: 0.000
Gravity	n: 1 — duration: 0.000
Centrifuge	n: 1 — duration: 0.000 — rotation speed: 0.000
Semi-Dynamic Pc>0	n: 1 — duration: 0.000 — Nonref Fluid rate: 0.000
Semi-Dynamic Pc<0	n: 1 — duration: 0.000 — Ref Fluid rate: 0.000
Customized	
Nonref Fluid and Ref Fluid at constant rates (or 0)	n: 1 — duration: 0.000 — Ref Fluid rate: 0.000 — Nonref Fluid rate: 0.000
Ref Fluid at constant pressure - Nonref Fluid at cst rate (or 0)	n: 1 — duration: 0.000 — Ref Fluid pressure: 0.000 — Nonref Fluid rate: 0.000
Nonref Fluid at constant pressure - Ref Fluid at constant rate (or 0)	n: 1 — duration: 0.000 — Nonref Fluid press: 0.000 — Ref Fluid rate: 0.000

Table 4: Block times entries.

Bond, Capillary, and Rapoport Numbers

When they can be calculated, the option to display the bond number, the capillary number, or the Rapoport number will appear on the Block Time window. Interfacial tension σ and wettability angle θ are required for the Bond or Capillary numbers.

The bond number is available for a centrifuge experiment; the capillary number for a one-fluid injected experiment, with an imposed pressure; and the Rapoport number for one-fluid injected with imposed flow rate.

The Bond number is defined as: $Nb = \frac{K <r> \omega^2 \Delta\rho}{\sigma cos\theta}$, with K the absolute permeability, <r> the distance from the axis of centrifuge to the middle of the sample, ω the centrifuge rotation speed, and Δρ the difference of density.

The Capillary number is defined as: $Nc = \frac{K\Delta P}{L\sigma cos\theta}$, with L the sample length and ΔP the difference in pressure ($P_{in} - P_{out}$) (see reference Dullien).

The Rapoport number is defined as: $N_R = L\ \mu\ v_i$, with L the sample length in cm, μ the viscosity of the fluid injected in cp, and υ_i the interstitial velocity in cm/min (the flow rate divided by the section divided by the porosity). The Rapoport number is not adimensional (Rapoport and Leas, 1953).

Inlet Face Capillary Pressure, P_C

For centrifuge experiments, a check box allows the calculation of the inlet capillary pressure, P_c. The calculation of Pc will occur once "Validate" is pressed, according to the formula:

$$P_C = \frac{1}{2}\Delta\rho\ \omega^2\ (R_{max}^2 - R_{min}^2),$$

with $\Delta\rho$ the difference of density, ω the centrifuge rotation speed, and R_{max} and R_{min} the interior and exterior distances to the rotation axe of each face of the sample.

View (curve display)

The View menu opens a window for displaying graphs. On the left-hand side, graphs can be displayed as a function of reference saturation, non-reference saturation, or liquid saturation (reference saturation + third phase saturation).

On the right-hand side, graphs can be displayed for reference data, non-reference data, or total.

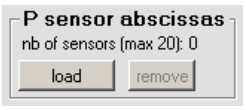
Figure 41: The "view" menu for selecting the graphs.

Local Pressures (version 2014)

There is the possibility to display the pressure versus time data $P(t)$ at different abscissas x along the sample with one graph per abscissas. The results of simulations at these locations can be compared to the measured values when pressure taps are available along the sample.

Loading the abscissas

Experimentally these data are obtained by placing pressure sensors along the sample. This is the reason why abscissa values are loaded in the window "type of experiment" (see also the "Type of experiment" section page 48) and are referenced in Cydar as "P sensor abscissas". Figure 42 shows the P sensor abscissas frame in the window "type of experiment".



no data loaded yet

3 sensor abscissas loaded

Figure 42: P sensor abscissas frame in the window "type of experiment" of the two-phase flow module.

The number of abscissas already loaded is displayed next to the label "nb of sensors (max 20):"on the right. This label also highlights that the maximum number of sensors is 20.

The left button has the caption “Load” if no data exist. In that case it opens a new spreadsheet to load a new data. If data already exist it has the caption “view/modify” and it opens a spreadsheet with the data. The “remove” button allows the user to erase data. It is enabled only if any data exist.

In the spreadsheet only the “X” column is loaded. So there is no need to enter “Y” values.

Remark: At the opening of a spreadsheet with existing data the second column is filled with ‘1’, this is for layout purpose. On the graph the data is on a line with ordinate y=1. Changing these values has no effect.

The ! is a warning displayed on a tooltip over it. This warning indicates that the P sensors abscissas are different that the abscissas loaded with pressure profiles. Then only simulation curves will be shown on the $P(t)$ graphs.

Graphs

For each sensor abscissas x there is a given graph which may display the experimental data and the simulation curves of the pressure versus time data at the given abscissa.

If any abscissas have been loaded a new frame is displayed on the right side of the “Graphs and Tables” window (Figure 43)

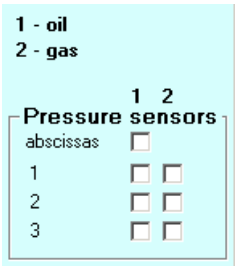


Figure 43: New frame on the "Graphs and Tables" window to open the graphs associated with the P sensors data.

The checkbox “abscissas” opens the graph displaying the abscissa values along the sample.

The numbered checkboxes correspond to the abscissas in ascending order from top to bottom. The checkbox “1” opens the graph displaying the $P(t)$ curves at the smallest abscissa, the checkbox “2” opens the graph associated with the abscissa just bigger, and so on. There are two columns. The first one corresponds to the reference fluid, the second one to the non-reference fluid.

Data points

The measured local pressures are calculated from the pressure profiles loaded in the “Load/fit data Window” (see page 45).

Simulation curves

The simulation curves can always be displayed on the graphs.

This is not the case for the “data points” curves. They can be displayed on the graphs only if there abscissas correspond to the sensor abscissas in number and in value with an error smaller than 10^{-3}. If this is not the case a warning is displayed on the frame “P sensor abscissas” with !. It displays on a tooltip a text explaining that reference and/or non-reference fluid “data points” cannot be displayed on the graphs (Figure 44).

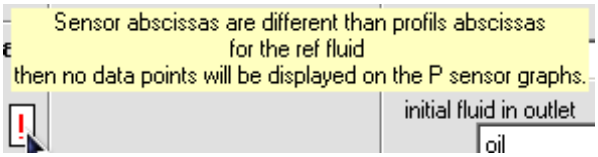


Figure 44: warning on the "P sensor abscissas" frame.

Displaying P sensor abscissas

The abscissas may be displayed as black vertical lines on the graphs associated with a data versus length, i.e. the profiles. For that purpose a checkbox “x P sensors” is placed on the “graph edition panel” in the tab “curves” (Figure 45).

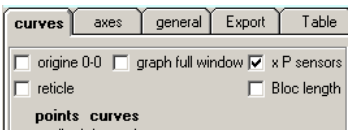
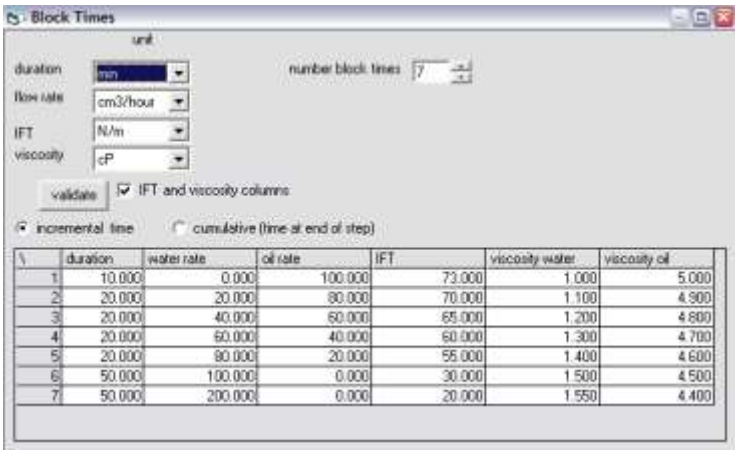


Figure 45: checkbox to display the abscissas of the pressure sensors on profile data.

Interfacial tension and viscosities values per block time (version 2016)

The purpose is to simulate a waterflood (imbibition) followed by injection of a surfactant or a polymer. Low surface tension or high viscosity increases the viscous forces compared to capillary forces and reduce the "capillary end effect". The change of fluid properties at end of simulation allows the quantification of recovery due to this effect.

It is possible to enter different IFT and viscosity values at each block times. It is first enabled in the block times window by checking the checkbox "IFT and viscosity columns". This displays the IFT and viscosity units comboboxes as well three more columns on the left (Figure 46).



\	duration	water rate	oil rate	IFT	viscosity water	viscosity oil
1	10.000	0.000	100.000	73.000	1.000	5.000
2	20.000	20.000	80.000	70.000	1.100	4.900
3	20.000	40.000	60.000	65.000	1.200	4.800
4	20.000	60.000	40.000	60.000	1.300	4.700
5	20.000	80.000	20.000	55.000	1.400	4.600
6	50.000	100.000	0.000	30.000	1.500	4.500
7	50.000	200.000	0.000	20.000	1.550	4.400

Figure 46: Block times window with the IFT and viscosities columns enabled.

Values

By default all the IFT column values are set to zero and the viscosities are set to the values entered in the "sample and fluids" window.

The IFT or viscosities change between BT are disabled in simulation if all BT values are equal. Therefore, if all the IFT values are zero, which is the default, the IFT change between block times is disabled in simulation.

The viscosities cannot be set to zero. If a zero is entered the value is set to its default value.

An error occurs if one of the values is zero or negative.

	IFT	**Viscosities**
Default values	0	Viscosity entered in the "sample and fluids" window.
Disabled	If all values are equal	If all values are equal
Error if enabled	One of the value is zero or negative	One of the value is zero or negative

Enabling the IFT and viscosities change between block times in simulation

The use of the values entered in the block times window is enabled by checking "BT-EOR" in the simulation window. This checkbox is enabled only if "IFT and viscosity columns" has been checked in the "Block Times" window.

If enabled the viscosity used in simulation is the one defined in the block times. Therefore, if the viscosity defined in the "sample and fluids" window is different to the first BT value the value in the sample window is not used for simulation.

IFT change implementation

The effect of the change of the interfacial tension is implemented only on the capillary pressure not on the relative permeability.

Using the assumption that the Leverett function is constant on the same rock type we can write that the Leverett function of the block time *i* is equal to the Leverett function of the block times *i+1*:

$$J_i = J_{i+1}$$

$$\frac{P_{c_i}}{\gamma_i \cos\theta_i}\sqrt{\frac{K}{\phi}} = \frac{P_{c_{i+1}}}{\gamma_{i+1} \cos\theta_{i+1}}\sqrt{\frac{K}{\phi}}$$

Therefore, we have between block time *i* and block time *i+1*:

$$P_{c_{i+1}} = P_{c_i}\frac{\gamma_{i+1}\cos\theta_{i+1}}{\gamma_i \cos\theta_i}$$

Remark: the IFT column contains the values $\gamma \cos\theta$ which have the same dimension than an interfacial tension.

Viscosity change implementation

The Darcy's law is:

$$U = \frac{K\,Kr}{\mu}\frac{\partial P}{\partial x}$$

The change of viscosity intervenes in the mobility factor $K\,Kr/\mu$. Therefore, to account for the change of the viscosity between two block times we just need to multiply the mobility factor by the viscosity ratio μ_i/μ_{i+1}.

Numerical Simulation

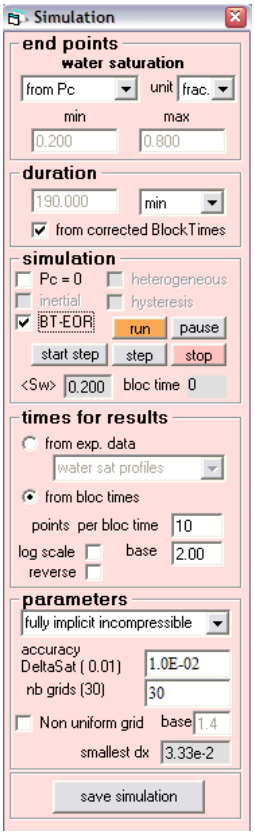


Figure 47: The simulation window of the two-phase flow module.

Principle of Simulations:

- Simulations are 1 dimensional,
- Space is discretized (grid blocks),
- Boundary conditions corresponds to the experimental conditions,
- Iteration in time starting from initial saturation,
- The numerical method is described in appendix.

End points:

Since experiments are generally performed on different samples, the end-points are not the same for Kr and Pc. However, the numerical simulations need the same values.
Selection of the reference fluid saturation end-points:

- From Pc
- From Kr
- Values chosen by the user.

Pc and Kr curves are rescaled within the chosen interval.

Simulation:

- "**Pc=0**": fixes the capillary pressure to 0.
- "**Inertial**": enabled when at least one density and one inertial parameter of a given fluid exist.
- "**Heterogeneous**": enabled when K and Phi data exist. When checked, allows to perform simulation with non-uniform K and phi.
- "**Hysteresis**": enabled when imbibition and drainage Pc curves exist for hysteresis.
- "**BT-EOR**": enable the possibility to change the interfacial tension IFT and the fluids viscosity at each block time.
- Buttons: "**run**", "**pause**", "**start step**", "**step**", "**stop**" the simulation.
- "**Block time**" output: number of blocks realized.
- "**<S_w>**" output: average saturation of the reference fluid.

Times for results: Number of simulation points shown on the graphic can be given from experimental data or a given number of points per block times. Abscissa can be on linear or log scale.

Parameters:

- Numerical scheme: "**fully implicit**" and "**fully implicit compressible**".
- "**Accuracy**" for the variation of saturation in the grid block (default is 0.01). A smaller value can be used when there is a message of no-convergence.
- "**Nb grids**": 30 is the default number, a good compromise between speed and accuracy when there are capillary effects that spread the saturation fronts. However, approximately 500 grids are necessary for illustration of Buckley-Leverett sharp front (without Pc).
- The "**smallest dx**" of the grid is a read-only value.

Optimization

The optimization module uses the routine BCNLS from the IMSL Library which solves a nonlinear least squares problem.

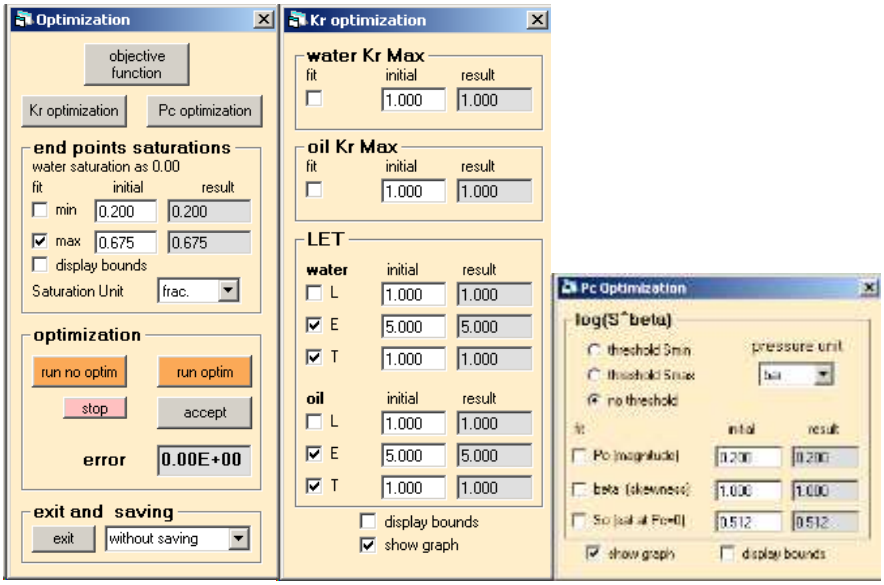


Figure 48: Optimization panels for Kr fitted with the LET function and Pc with a function Log(beta).

The optimization module is an automated tool for history matching of the relative permeabilities and capillary pressure curves.

For optimization, Kr and Pc curves must be fitted with an analytical curve. Optimization with Corey (power law), modified Corey's model (modified power law), or LET function for Kr; Log(beta) model for Pc, and interpolations for both have been implemented. Optimization per points is also possible for Kr and Pc (see below).

Optimization is a difficult process, especially with a large number of parameters. The main recommendation is to optimized a minimum number of parameters at a time, and choose the objective function using physical considerations. For instance, a final saturation will be sensible to a production variable, and a Kr value to a pressure drop. An example is given in the tutorial below.

Each parameter can be selected using the corresponding box; the user assigns an initial value and CYDAR provides a final value. To run an optimization, click "**run optim**" on the main panel (left window Figure 48) and valid the optimization with the "**accept**" button.

Lower and upper bounds for each parameter can be changed, using the "**display bounds**" checkbox on the corresponding window (Figure 49).

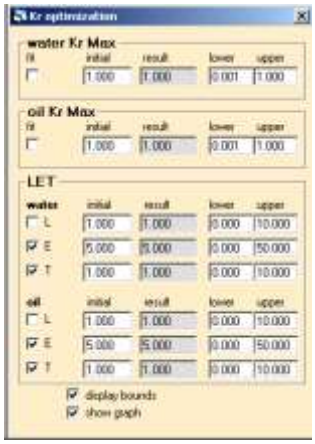


Figure 49: displaying bounds.

Objective function

The optimization can be done on the effluent volumes, the average saturation, the pressure drop, or the pressure and saturation profiles. If several objective functions are selected, as in Figure 50, the user can impose different weights.

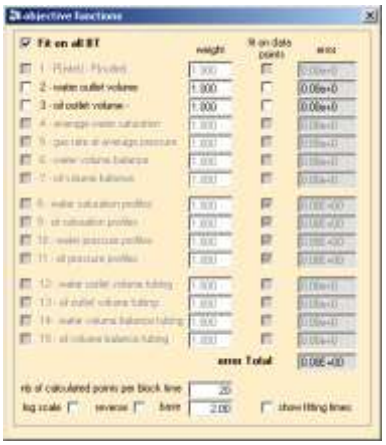
(a)  (b) 

Figure 50: Define the objective function by selecting experimental data to be compared to results of numerical simulation.

The optimization can be done on all block times (as shown top left Figure 50a) or on selected block times (Figure 50b).

The user can choose the number of calculated points per block time and their position on linear or log scale (bottom Figure 50a). These later entries are the same as in the simulation window. The "**show fitting times**" checkbox displays the simulation time abscissas distribution along the time axis. For selected data, the objective function can be calculated on data points ("**Fit on data point**" Figure 50a).

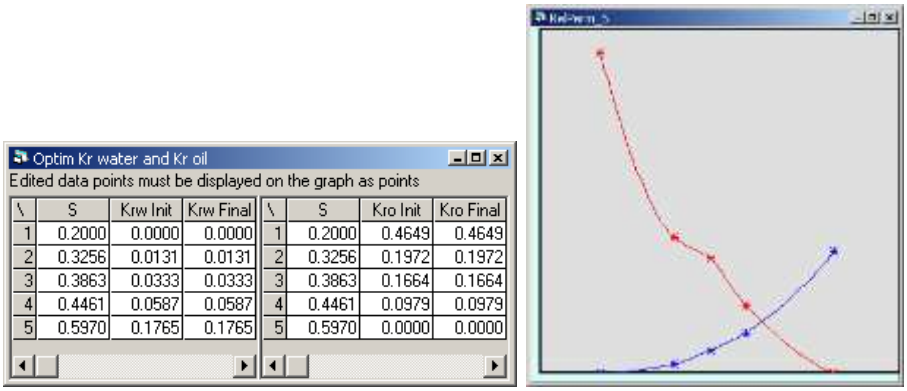
The **"weight"** allows giving more weight on one function. This option is particularly useful when optimizing on two production volumes, when there is a large difference between the two volumes (for instance, oil production is ~5 cc and water production is ~500 cc). "Weights" can be larger than one; the ratio of all weights is then calculated.

Optimization per points

If Kr or Pc are interpolated using splines, Akima, or Hermite interpolations, they can be optimized per points.

Moving points

Clicking on the "**Kr optimization**" or "**Pc optimization**" buttons displays the corresponding graphs, and a table with the interpolation knots coordinates, i.e. the data points (Figure 51).



\	S	Krw Init	Krw Final
1	0.2000	0.0000	0.0000
2	0.3256	0.0131	0.0131
3	0.3863	0.0333	0.0333
4	0.4461	0.0587	0.0587
5	0.5970	0.1765	0.1765

\	S	Kro Init	Kro Final
1	0.2000	0.4649	0.4649
2	0.3256	0.1972	0.1972
3	0.3863	0.1664	0.1664
4	0.4461	0.0979	0.0979
5	0.5970	0.0000	0.0000

Figure 51: Interpolation knots table and the corresponding Kr graph.

Moving points can be done either by drag-and-drop on the graph or by changing the values in the table.

Edited data points must be displayed on the graph as points in order to move them by drag-and-drop. The drag and drop is an intuitive operation: place the mouse cursor on a point, left click, move the point, release the mouse button and drop. The value is automatically updated in the table.

In the table, the first (saturation) and second (Pc or Kr) column can be changed, the graph being updated automatically.

Moving action can be undone in two ways:

- Through a right click menu which allows:
- Three actions for Kr: undo all moves for one or both Kr (Figure 52).
- One action for Pc "undo all moves".
- Undo all moves performed since the start of the optimization. This operation cannot be undone.

Ctrl+Z which undoes only the last move if no other action has been performed. When done once, Ctrl+Y allows a redo action for the same move. It applies on the last action performed either on the graph or in the table.

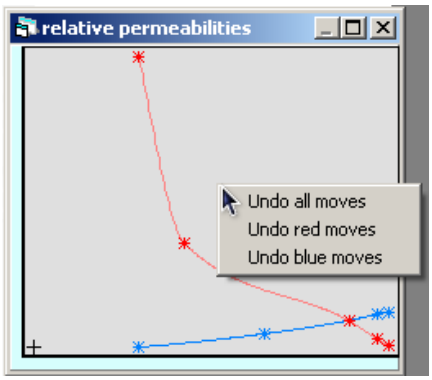


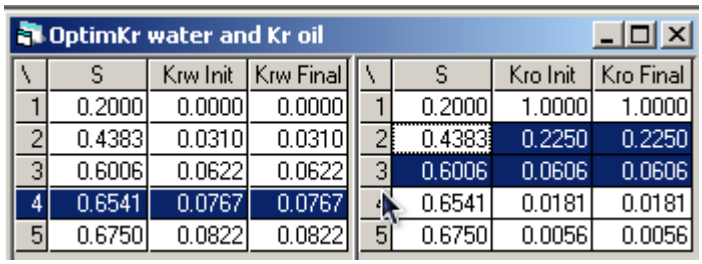
Figure 52: Right click popup in an optimization per points.

Optimization

Points to be optimized are selected and highlighted by clicking on the row number, i.e. the left column of each table. The different ways to select row are:

- One click selects the point.
- To select a range of points just hold down the left mouse button move the cursor over the range.
- If the Ctrl key is held down, each clicked row is selected or unselected.

When an optimization is run, the entire tables are locked.



\	S	Krw Init	Krw Final
1	0.2000	0.0000	0.0000
2	0.4383	0.0310	0.0310
3	0.6006	0.0622	0.0622
4	0.6541	0.0767	0.0767
5	0.6750	0.0822	0.0822

\	S	Kro Init	Kro Final
1	0.2000	1.0000	1.0000
2	0.4383	0.2250	0.2250
3	0.6006	0.0606	0.0606
4	0.6541	0.0181	0.0181
5	0.6750	0.0056	0.0056

Figure 53: Optimization per points; selection of points

Format number can be changed by column through the right click menu on the table or the shortcut Ctrl+F.

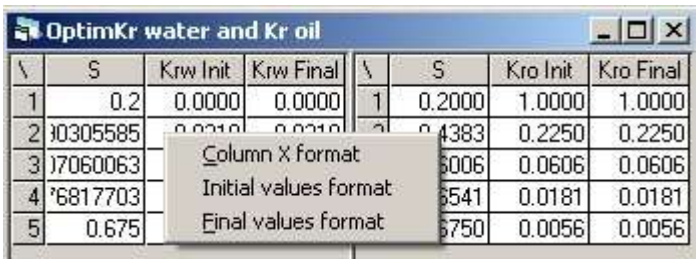


Figure 54: In the optimization table, a right click popup allows changing format of numbers per column.

Remarks:

- Kr and Pc data cannot be edited during an optimization session.
- Before saving the project, it is recommended to exit the session with the "exit" button, and to choose either if "no curve", "Kr", "Pc" or both have to be saved.
- An example of optimization is given in the unsteady-state section.

Unsteady state

Analytical methods (JBN)

For a "**1 fluid injected**" experiment, the "**analytical JBN**" button is enabled in the "**calculation**" frame of TPF main window (Figure 33), and the "**JBN-Jones and Roszelle**" window can be open (Figure 55).

Window JBN-Jones and Roszelle

Injection frame:

- Type of displacement: drainage or imbibition
- Type of injection: Constant flow rate or constant pressure

Breakthrough time:

The breakthrough time must be given because calculations are only valid when the two fluids are produced. Corresponding volume is an output, for verification with the experimental value.

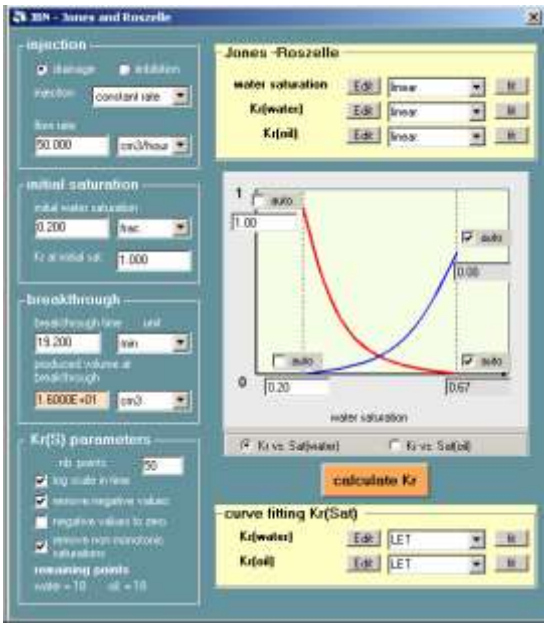


Figure 55: Window for JBN calculation.

Kr(S) "parameters":

The user can specify the number of points and several constrains such as removing negative values.

Jones-Roszelle:

Smoothing the intermediate curves (Kr and saturation as a function of time) is equivalent to the Jones and Roszelle method. Using "**linear**" fitting leads to the original JBN method.

The schematic display of the Kr curves allows changing the end-point values if necessary. These curves can be displayed either versus water saturation or oil saturation.

Curve fitting:

The curve fitting part gives the same tools than the Fit section of "**Data points**". The final calculated Kr points can be fitted either using the JBN window or the general window for data editing.

IMPORTANT: JBN calculation assumes that the capillary pressure is negligible. The standard method to determine the Kr must introduce the measured or estimated Pc, and use optimizations (see the corresponding section).

Multistep one fluid injected

The JBN method is valid only with the unsteady state experiment, with either a constant flow rate or a constant pressure.

In a first step, saturation and relative permeabilities are calculated as functions of time from the effluent volumes and pressure drop measurements. Then the time is eliminated and Kr are given as a function of saturation.

The different methods called JBN (from Johnson, Bossler and Neuman), Welge, or Jones and Roszelle are based on the same analytical calculation.

If, due to experimental noise, saturation is a non-monotonic function of time, then several different values of Kr can correspond to the same value of saturation (Figure 56). This issue can be avoided by using the Jones and Roszelle method, which smooth the saturation and Kr as a function of time before eliminating the time variable.

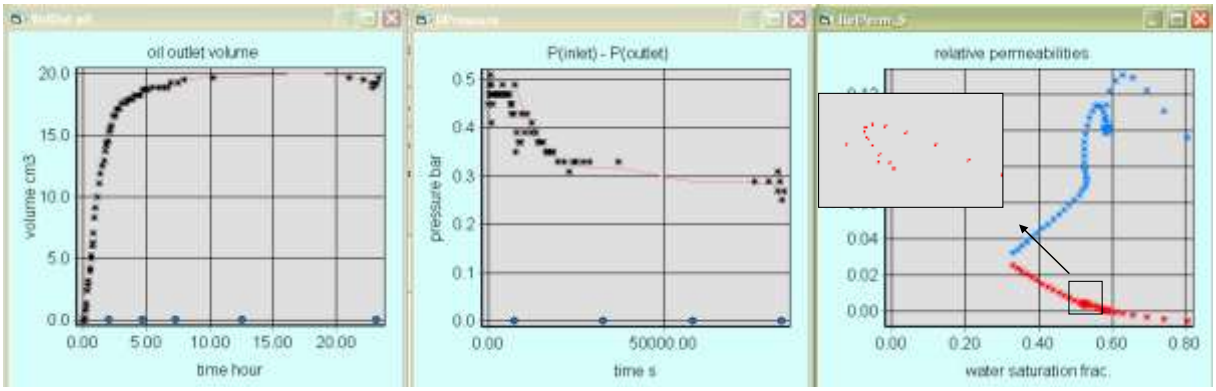


Figure 56: Example of Kr determination with noisy data leading to several Kr values.

Tutorial Kr_USS_MW

This is a synthetic case of water/oil imbibition where the experimental data are calculated using a direct numerical simulation; then the parameters are determined and compared to the initial values.

The tutorial contains several files placed in the folder KrUSS:

- **Tutorial_Kr_USS_MW.xls:** Microsoft Excel file with the experimental data, sample, fluids, production, and pressure drop.
- **Tutorial_Kr_USS_MW_direct.cyd:** file used to create "experimental data".
- **Tutorial_Kr_USS_MW.cyd:** starting a CYDAR project with the sample and fluids properties, experimental conditions, block times values, experimental data (average saturation and delta pressure). There is no Pc or Kr.
- **Tutorial_Kr_USS_MW_JBN.cyd:** analytical calculation of Kr done from experimental data, Kr fitted with power law (Corey) for optimization, and analytical Pc.
- **Tutorial_Kr_USS_MW_optim_initial.cyd**. This file is the starting point for numerical optimization. For this purpose, the JBN Kr has been fitted with the Corey functions to allow parameter adjustments.
- **Tutorial_Kr_USS_MW_optim_final.cyd**: final results after optimization.

Starting a project

First, start a new project and choose "Kr unsteady state"; the default experimental type is set as "1 fluid injected". Then, enter the data:

- general information "**info**", sample and fluids properties "**sample/fluids**";
- experimental conditions (here most of them are already set) "**experiment**": type set to "1 fluid injected"; displacement set to "imbibition"; water injection type is "imposed flow rate", and initial saturation is 0.2.
- "block times";
- experimental data: "**Load/fit Data**", and after selecting the proper raw data type, "**load data points**";

At this stage, the project should be equivalent to Tutorial_Kr_USS_MW.cyd. If the "**numerical simulation**" button is pressed, an error message is displayed saying that no relative permeabilities are set.

Kr calculation

The calculation is only valid after the breakthrough time; therefore data are fitted with splines only from this time.

To open the "JBN" window, click on the "**analytical JBN**" button in the TPF home window. Flow rate and initial saturation should already be set to their proper values. Enter the breakthrough time and press "**calculate Kr**".

Results are displayed in the Microsoft Excel file. The important and well known result is that the JBN calculation does not produce the right Kr, since the capillary pressure is not taken into account. The main difference is the final saturation, lower than the real one (over-estimation of residual oil saturation, see paper by Shell in SCA). The exact final saturation is S_w=0.72 that can be observed at the entrance of the sample in the saturation profiles curve. The JBN analytical calculation uses the average saturation by assuming a uniform saturation (no capillary pressure), corresponding to a value of S_w=0.679.

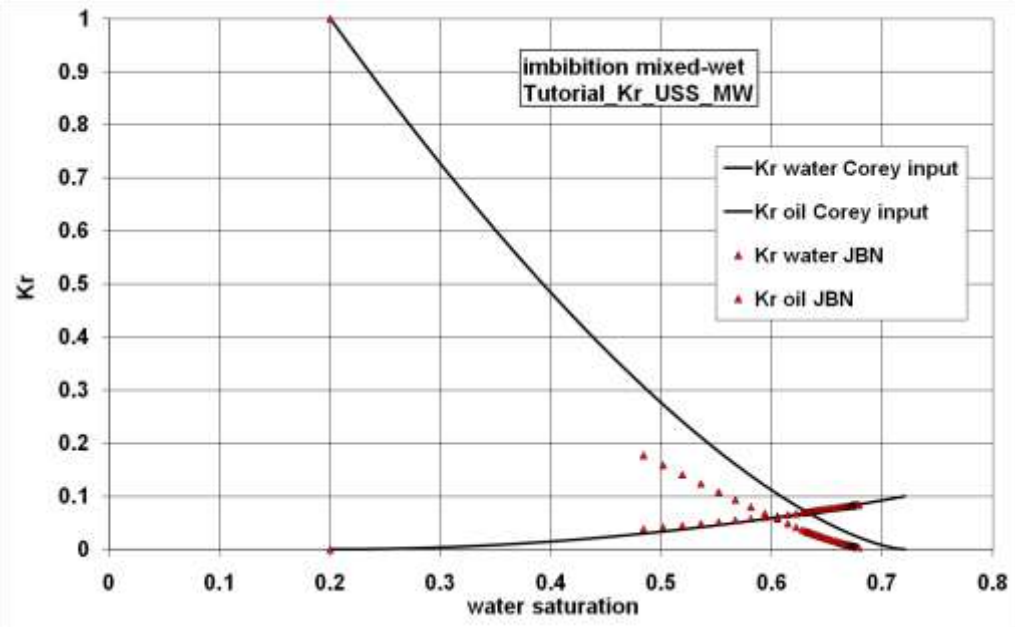


Figure 57: Example of Tutorial_Kr_USS_MW, input Kr (exact) and results of JBN calculation.

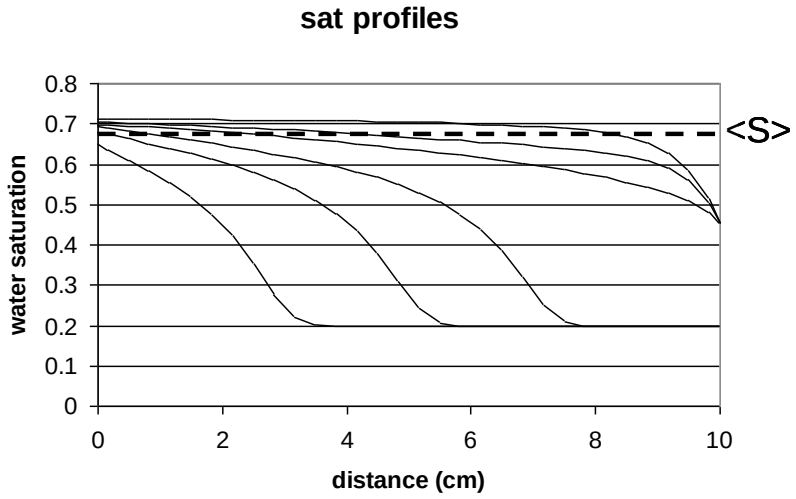


Figure 58: Example of Tutorial_Kr_USS_MW. The oil retention at the outlet is due to the negative part of the capillary pressure curve.

Simulation with initial guess

There is no Pc function entered yet; therefore, in the Simulation window, Pc is set to zero (checkbox "**Pc=0**" in the "simulation" frame). The end points must be set to "from Kr" at the top of the window. Now run a simulation.

Output Kr are not appropriate yet because the real case was done with a capillary pressure. We now need to enter the capillary pressure data.

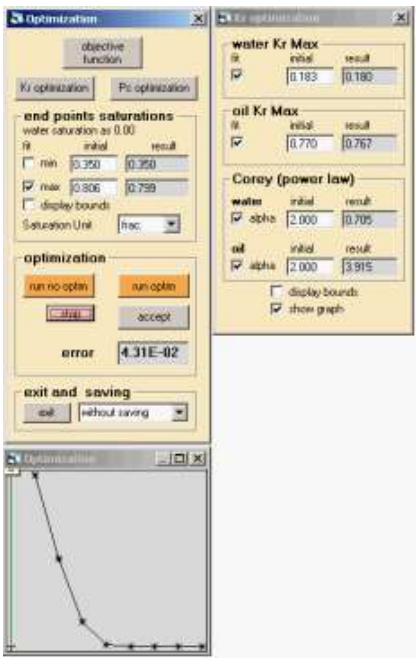
By default, the data will be linearly interpolated, but a Hermite interpolation gives a smoother curve. If a simulation is run with the linear interpolation, bumps due to the derivative change at the knots might be observed.

Now, performing a simulation with this Pc (unselect "**Pc=0**") shows that Kr needs optimization.

At this stage, the project is equivalent to Tutorial_Kr_USS_MW_JBN.cyd.

Kr optimization

First Kr curves should be fit with a power law. This optimization case works very well; therefore an optimization on the five parameters (the final saturation and the four Kr parameters) can be successfully performed. And results will be close to the original one: Smax=0.72, Kr water max = 0.10, n water=1.94, Kr oil max = 0.99 and n oil=1.49.

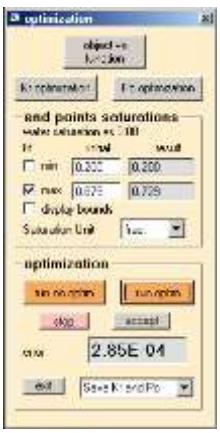


- Opening the "optimization" window: button "**optimization**";
- Selecting the data to fit and the "block times" on which to perform optimization: button "**objective function**", here keep "fit on all BT" and select data 1 and 3;
- Select Smax;
- Selecting Kr parameters: button "**Kr optimization**", select all parameters;
- "run optim".

At this stage, the project should be equivalent to Tutorial_Kr_USS_MW_Final.cyd

In most cases, an optimization cannot be performed on all parameters at once, and should be tried step by step. To perform a step by step optimization:

- The initial point for Kr is assumed to be perfectly known from the measurement at the end of first drainage (Oil Krmax and Swi).



Adjusting the final saturation: corresponding to the end points max, in the optimization window.

- With "run no optim", we can verify that increasing the final saturation to 0.7 gives a better fit.
- To do it automatically, open the window "objective function" and select "effluent oil volume", since this parameter is the most sensible to the final saturation.
- select the max saturation
- click "run optim",

The optimization reaches a bound for S=0.729. Accept the value (button "**accept**") and start again the optimization. To converge, the initial value must be close to the optimized one. Accept this value.

The optimization graph shows the change in the error during the simulation. When a stable value is obtained on this curve, the optimization can be stopped using the stop button. Wait for the complete stop of the process before accepting the value (the mouse index changes from the hour-glass to the arrow). The simulation can also be stopped using the stop button in the simulation window or using the ESC key on the keyboard.

The message "the fit cannot be improved" is a normal message that indicates that the optimization process has found a value that minimizes the objective error.

Adjusting water Kr max

We observe that the simulated pressure (DP) in red is higher than the experimental one. At the end of the experiment, oil is no longer produced and water is mainly flowing by itself. Consequently, the final pressure drop is mainly controlled by the relative permeability of water and roughly proportional to the krmax value.

To adjust this parameter automatically, check the fitting box and select delta pressure in the objective function. Then "**run optim**". The fit gives *Krmax* = 0.1.

Adjusting the Kr change the production. We can redo the optimization for the final saturation to have a better fit.

Adjusting both water Kr max, its Corey exponent, and final saturation

Now that the values are close to the optimum, all three parameters can be adjusted together.

Check the three boxes for fit and select both oil volume and pressure difference in the objective function (weight can be kept to unity for each variable). The fit is not so much improved.

Adjusting all parameters

We can optimize all five parameters together. That gives a very good fit with values close to the originals: Smax=0.72, Kr max water = 0.10, n water = 2.01, Kr oil max = 1.00 and n oil = 1.4.

Steady-state

Analytical

The use of this module requires using either the average saturation or the effluent volumes. This method assumes a uniform saturation along the sample, an assumption that is never valid due to the presence of capillary forces. The result must be considered as a first guess for a more accurate method taking into account the capillary pressure (numerical optimization).

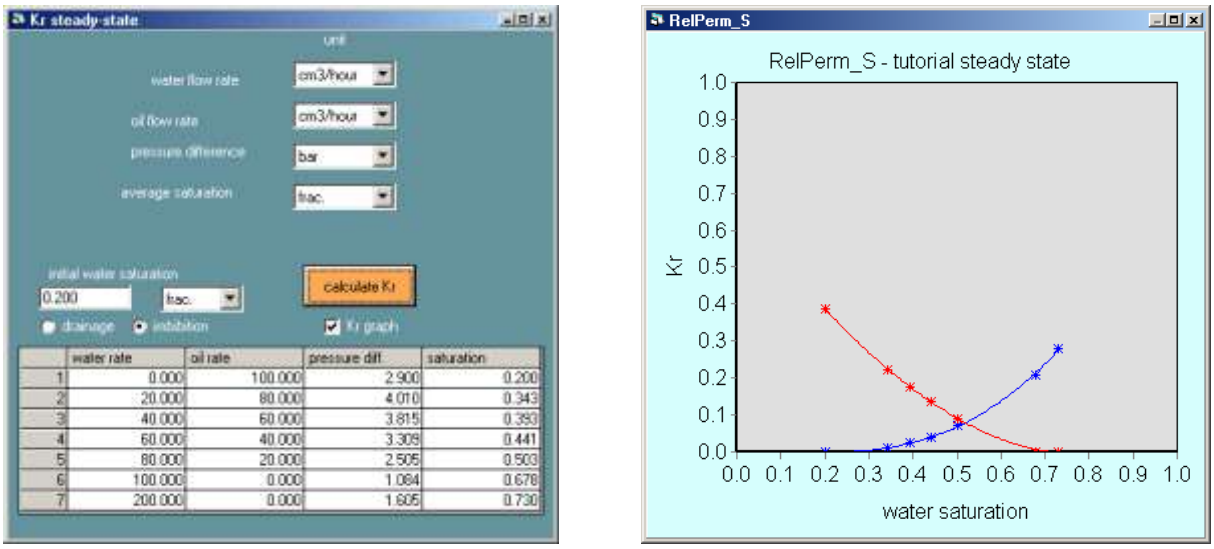


Figure 59: Kr steady-state window and the corresponding Kr result.

Calculation steps:

- Selecting the proper units for the flow rates, pressure, and saturation;
- Entering the initial saturation. By default this value is the one entered in the experiment window;
- Entering the flow rate columns. They are the same as the bloc times columns, unless the user does not want to perform calculation on all bloc times. Bloc times are not necessary.
- Pressure diff. and saturation columns are the equilibrium values on each step, or, if equilibrium was not reached on each step, the value at the end of step.
- Pressing "**calculate Kr**" runs the calculation.
- Check the box "Kr graph" (or menu view) to open the graph and to see results.

> **Useful Tip:** If after calculation, the graph does not show any data, open the graph properties window and select the curves to be displayed.

Tutorial

This is a synthetic case of water/oil imbibition where experimental data are calculated using a direct numerical simulation, and parameters are determined and compared to the initial values.

The tutorial contains several files placed in the folder SteadyState:

- **Tutorial_Kr_SteadyState_OW.xls:** Microsoft Excel file with the experimental data, sample, fluids, production, and pressure drop.
- **Tutorial_Kr_SteadyState_OW_direct.cyd:** file used to create "experimental data".
- **Tutorial_Kr_SteadyState_OW.cyd:** CYDAR project with sample and fluids properties, experimental conditions, the block times values, and experimental data (average saturation and delta pressure). There are no Pc and Kr yet.
- **Tutorial_Kr_SteadyState_OW_Kr.cyd:** analytical calculation of Kr done from experimental data, Kr fitted with power law (Corey) for optimization, and analytical Pc.

Starting a project

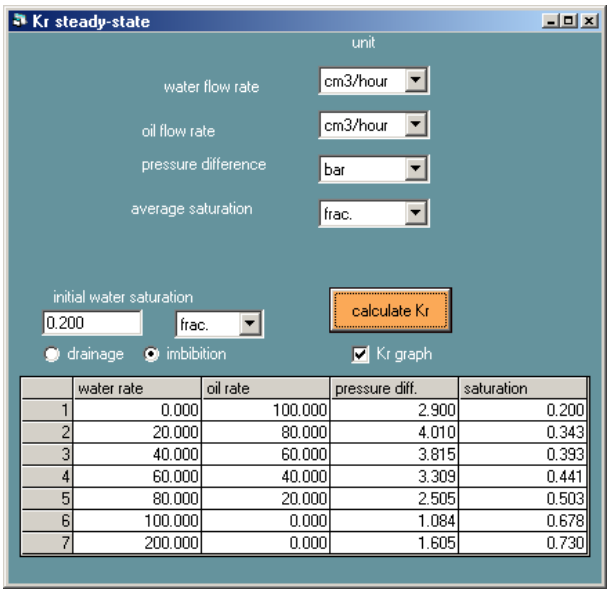
First, start a new project and choose "Kr steady state". This sets the default experimental type as "2 fluids injected". Then, enter the following data:

- general information "**info**", sample, and fluids properties "**sample/fluids**";
- experimental conditions (here most of them are already set) "**experiment**": type set to "2 fluids injected", displacement set to "imbibition", water and oil injection type are "imposed flow rate", and initial saturation is 0.2.
- "block times";

- experimental data: "**Load/fit Data**" then, after selecting the proper raw data type, "**load data points**";

At this stage the project is equivalent to Tutorial_Kr_SteadyState_OW.cyd. If the "**numerical simulation**" button is pressed, an error message is displayed saying that no relative permeabilities are set.

To open the "Kr steady-state" window, click on the "**analytical steady-state**" button in the TPF home window.



	water rate	oil rate	pressure diff.	saturation
1	0.000	100.000	2.900	0.200
2	20.000	80.000	4.010	0.343
3	40.000	60.000	3.815	0.393
4	60.000	40.000	3.309	0.441
5	80.000	20.000	2.505	0.503
6	100.000	0.000	1.084	0.678
7	200.000	0.000	1.605	0.730

- Select the proper units;
- The initial water saturation was set in the experiment window;
- The displacement type was set in the experiment window;
- Enter data: a copy and paste from block times gives the flow rate columns; the "pressure diff." and "saturation" values are equilibrium values. The first five steps seem close to equilibrium state. The last two steps are clearly not at equilibrium. However, we will still take experimental values at end of steps.
- "**calculate Kr**" performs calculation.

At this stage, fit the raw data with a power law (Corey):

- open the "**Data points**" window (button "**Load/fit Data**");
- select the Kr (options "**Kr water**" or "**Kr oil**");
- select the "**power**" fit and click "**fit**".

Simulation with first guess

No Pc function have been entered yet, therefore the "Simulation" window Pc is set to zero (checkbox "Pc=0" in the "simulation" frame). The end points must be set to "from Kr" at the top of the window.

Now run a simulation. The results look good, except for the final saturation. However, these Kr values do not include capillary pressure. Now enter the analytical Pc used to create "experimental data":

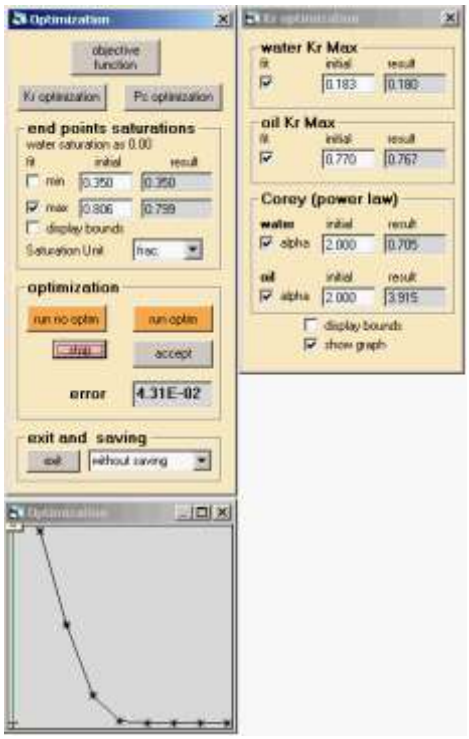
- Open the "capillary pressure" window: button "**Pc input**";
- Select the analytical function "**Log(S_beta)**": end points 0.2-0.8, "threshold Smin", Po=0.1 bar, Pt=-0.1 bar.
- Press "**calculate**"

Performing a simulation with this Pc value (unselect "**Pc=0**") shows that Kr need optimization.

At this stage, the project should be equivalent to Tutorial_Kr_SteadyState_OW_Kr.cyd.

Kr optimization

This optimization case is ideal; therefore an optimization with the five parameters (the final saturation and the four Kr parameters) gives the expected values (S_{final}=0.8, Kr_{max} water and oil = 0.4, n_{water}=2.5 and n_{oil}=2).



- Open the "optimization" window: button "**optimization**";
- Select the data to fit and the "**block times**" on which to perform optimization: button "**objective function**", here keep "**fit on all BT**" and select data 1 and 4;
- Select S max;
- Select Kr parameters: button "**Kr optimization**", select all parameters;
- "**run optim**".

At this stage, the project is equivalent to Tutorial_Kr_SteadyState_OW_Optim_Final.cyd.

Useful Tip: Most of the time, an optimization must be performed step by step as shown in paragraph "Tutorial Kr_USS_MW".

Centrifuge

Historically, centrifuge has been used to measure Pc curves in positive drainage and negative imbibition. Now it is also used to determine parts of the relative permeability curves in single or multi-speed experiments using numerical optimization.

Local Pc from average

The main window is accessible either from CYDAR main window button "analytical Pc Forbes" or from the capillary pressure window button "**from centrifuge**". Data can be entered as saturation (not local) versus Pc or saturation versus speed. In each case, CYDAR automatically provides the second one.

There are three analytical methods to calculate the local Pc curve from the average Pc curve (option choice frame "**method**" (Figure 60)):

- "**Hassler-Brunner**" or HB.
- "**Forbes continuous**" uses Forbes' analytical development but not its discretization method. Calculations are performed on the analytical fit of the data and are more accurate.
- "**Forbes + splines**" optimization.

There is one quality control tool:

- "**Pc(<S>) from Pc(S)**": calculation of the average Pc curve from the local Pc (see section "Tutorial Pc calculation: Comparison CYDAR – Forbes SCA"). It allows to check a local Pc curve comparing experimental data and calculated Pc(<S>) curve.

Optimization tool

CYDAR provides an automatic tool to optimize the local Pc curve Pc(S), such as the calculated average curve Pc(<S>) fits better the experimental average data. The Forbes continuous calculation is used as a first guess.

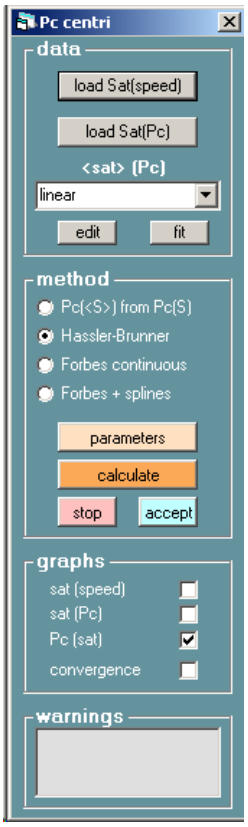


Figure 60: Pc centrifuge.

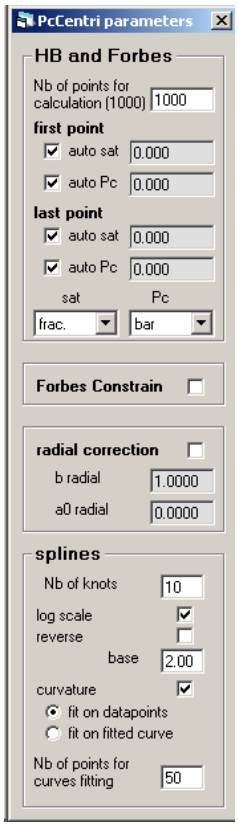


Figure 61: Parameters for the Pc centrifuge calculation

The user can control several parameters (Figure 61).

- Frame "**HB or Forbes**": end points of the local Pc curve, number of points for numerical integration of the average saturation (default 1000).
- Frame "**Forbes**": constrains on Forbes calculation imposes monotonicity of the local saturation versus pressure.
- Frame "**splines**": number of knots for splines correction (default between 3 and 5), log scale for the knots distribution, fit on data points or fit on the smoothed curve (with number of points, default 50).

Important notes on using "Forbes + Splines":

Because of the difference in the shape of the capillary pressure curve in imbibition and drainage, the position of the knots for calculation of the splines function usually needs to be adjusted as following:

For a drainage: In the parameter window, the reverse log scale parameter should be unchecked.

For an imbibition: In the parameter window, the reverse log scale parameter should be checked.

Tutorial Pc calculation: Comparison CYDAR – Forbes SCA

Files:

- Tutorial_PcCentri_Drainage_analytical.xls
- Tutorial_PcCentri_Drainage_analytical_simulation.cyd
- Tutorial_PcCentri_Drainage_Cydar_Optim.cyd
- Tutorial_PcCentri_Drainage_Forbes_5_Pts.cyd

In the following section, Pc(S) refers to local capillary pressure curve and Pc(<S>) refers to the capillary pressure versus averaged saturation curve.

The goal of this tutorial is to show the power of the CYDAR's optimization tool to calculate local Pc curve from experimental Pc(<S>) curve.

To test and compare the different calculation methods, we consider an analytical case where the local and the average Pc curves are well known. This case was presented by Chen and Ruth (J. Petr. Sc. Eng. (9) 1993):

$$S(P_{c1}) = \begin{cases} \dfrac{1.5}{P_{c1}} + 0.25 & \text{when} \quad P_{c1} \geq 2 \\ 1 & \text{when} \quad P_{c1} \leq 2 \end{cases}$$

The mean saturation can be calculated from:

$$S(P_{c1}) = \frac{1+\sqrt{1-B}}{2} \int_0^{P_{c1}} \frac{S(P_c)}{\sqrt{1+B\frac{P_c}{P_{c1}}}} dP_c$$

where $P_{c1} = \frac{1}{2}\Delta\rho\omega^2(r_2^2 - r_1^2)$, $B = 1-(r_1/r_2)^2$, r_1 and r_2 are the inner and outer radii, $\Delta\rho$ is the densities difference and ω is the rotation speed.

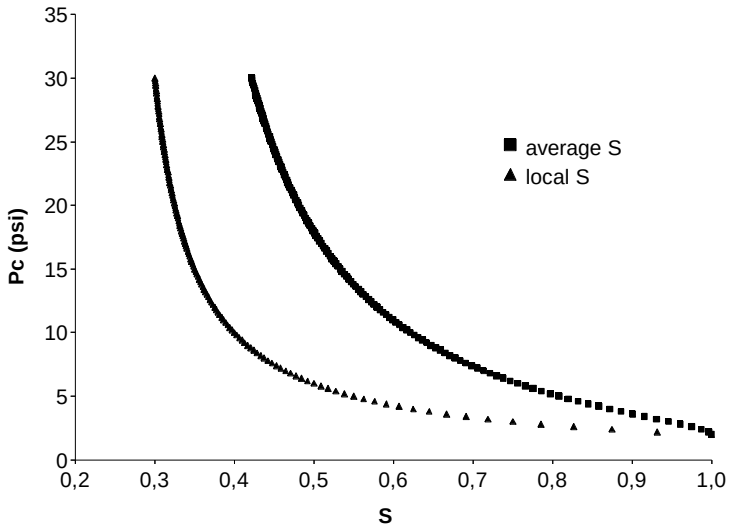


Figure 62: Theoretical Pc(S) and Pc(<S>) curves, analytical case from Chen and Ruth (1993)

Figure 62 shows the analytical Pc(S) and Pc(<S>) curves. We will now compare the results of the Pc(S) curve calculation from the Pc(<S>) curve, with the popular Forbes method as described in the SCA paper number 9107 and the CYDAR optimization tool.

Both methods use the second kind solution developed by Forbes (equations 15 and 20, Forbes SCA 1991), the difference comes from the treatment of it. In the 1991 SCA, Forbes develops a discrete method, which could be seen as a finite difference method (equations 18 and 21 Forbes SCA 1991). Therefore, if the Pc(<S>) curve has 20 points, calculation (and so derivatives) will be done with only 20 points. If the Pc(<S>) curve has only 5 points, the calculation is done with 5 points.

CYDAR optimization tool uses the same continuous equations (equations 15 and 20 Forbes SCA 1991) but uses spline fit or interpolation to perform calculation. The optimization is done to obtain the best fit between numerical Pc(<S>) curve (re-calculated from the Pc(S) curve) and the experimental Pc(<S>) data.

The results can be different when the number of points is 5 or lower. The discrete Forbes method uses a linear interpolation of the experimental data whereas CYDAR tool uses a more physical fit.

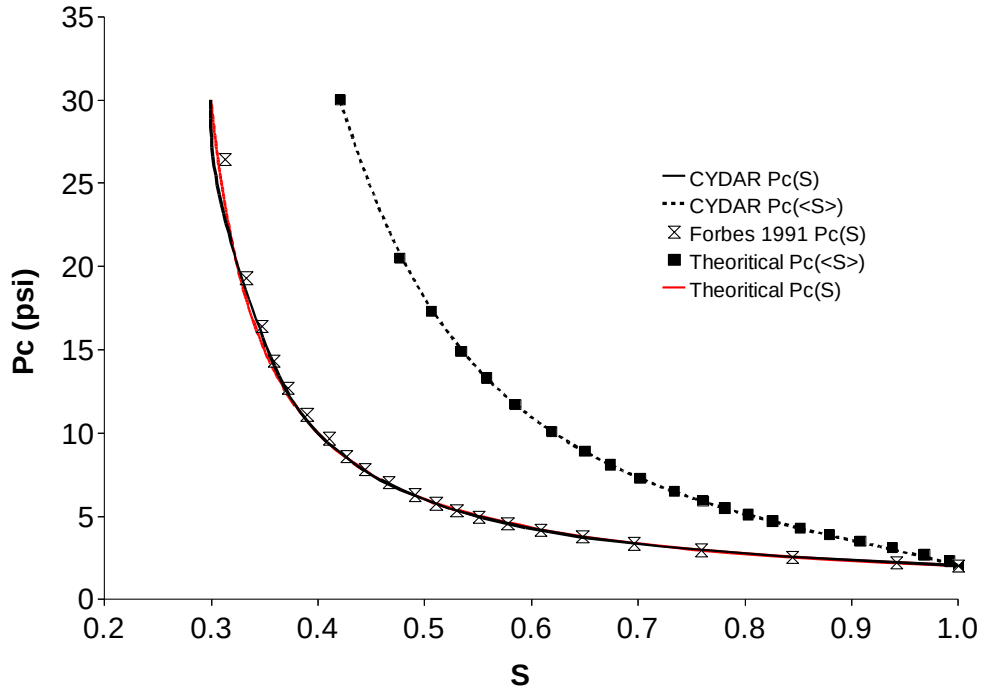


Figure 63: Analytical case with 23 points, comparison with Forbes SCA 1991 and CYDAR optimization tool.

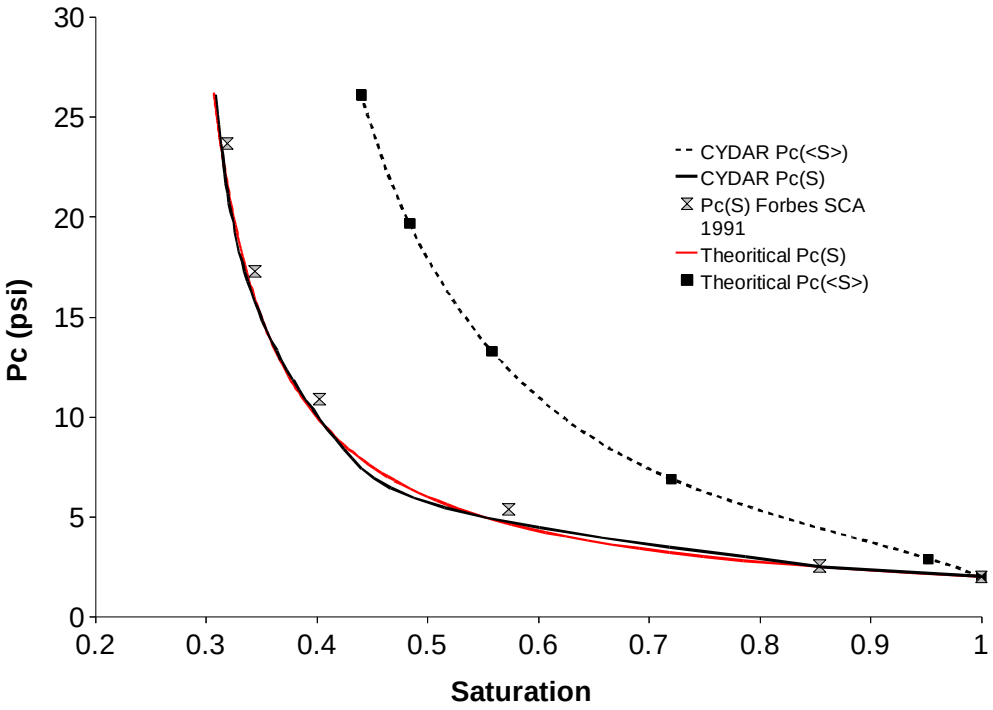


Figure 64: Analytical case with 6 points, comparison with Forbes SCA 1991 and CYDAR optimization tool.

Figure 63 shows results using a data of 23 points as original experimental data. Both methods give good results. The fits are good with the analytical Pc(S) curve (red line Figure 63) and with the Pc(<S>) curve (black squares Figure 63). The re-calculated Pc(<S>) curve in the SCA 1991 Forbes method will always fit exactly the original curve because it simply does a reverse calculation.

Figure 64 shows the same analysis, but with only 6 points as original data. The CYDAR tool still gives good results. In this case, a Hermite interpolation was used for the CYDAR optimization. But the Forbes discrete method has been shifted from the analytical Pc(S) curve.

All these results are summarized in the file Tutorial_PcCentri_Drainage_analytical.xls.

The difference in terms of production needs to be investigated. Using the analytical local saturation Pc(S) curve, we run a simulation with Corey as Kr and the experimental conditions shown Figure 65. Experimental setup and Kr are not critical here because we are interested in the average saturation plateaus which are controlled by capillary forces. The corresponding CYDAR file is Tutorial_PcCentri_Drainage_analytical_simulation.cyd.

This simulation is now used as an experimental data. From it we can enter the saturation versus speed data which counts 6 points, and using the two methods the local saturation Pc curve can be calculated. The corresponding CYDAR file is "Tutorial_PcCentri Drainage_Cydar_Optim.cyd."

We open the window "**Pc Centri**" (see above Figure 60):

- Select the "**Forbes + splines**".
- Click on the "**calculate**" button.
- Click the "**accept**" button to load this Pc curve.

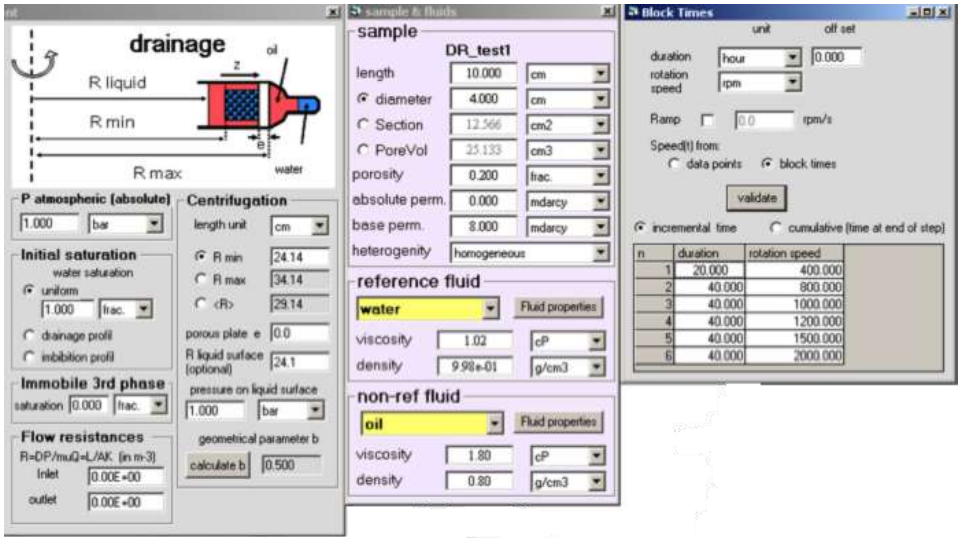


Figure 65: Simulation using analytical local saturation Pc curve.

Now we are able to run a simulation with the Pc(S) from Forbes SCA 1991 calculation, Pc(S) from CYDAR and with the analytical Pc(S) considered as the real Pc(S). The file "Tutorial_PcCentri Drainage_Forbes_5_Pts.cyd" has the Forbes' Pc(S) loaded; experimental data for mean saturation come from the simulation with analytical Pc(S).

Figure 66 shows the simulation results. In the graph the black stars are the experimental data (simulation using analytical Pc), the blue line is a simulation with the local Pc curve calculated with the Forbes discrete method, and the red line is the simulation using the CYDAR optimization tool. The results speak for themselves. The approximation made with the discrete method is not correct.

This is the main reason why the discrete Forbes SCA 1991 method is not implemented in CYDAR.

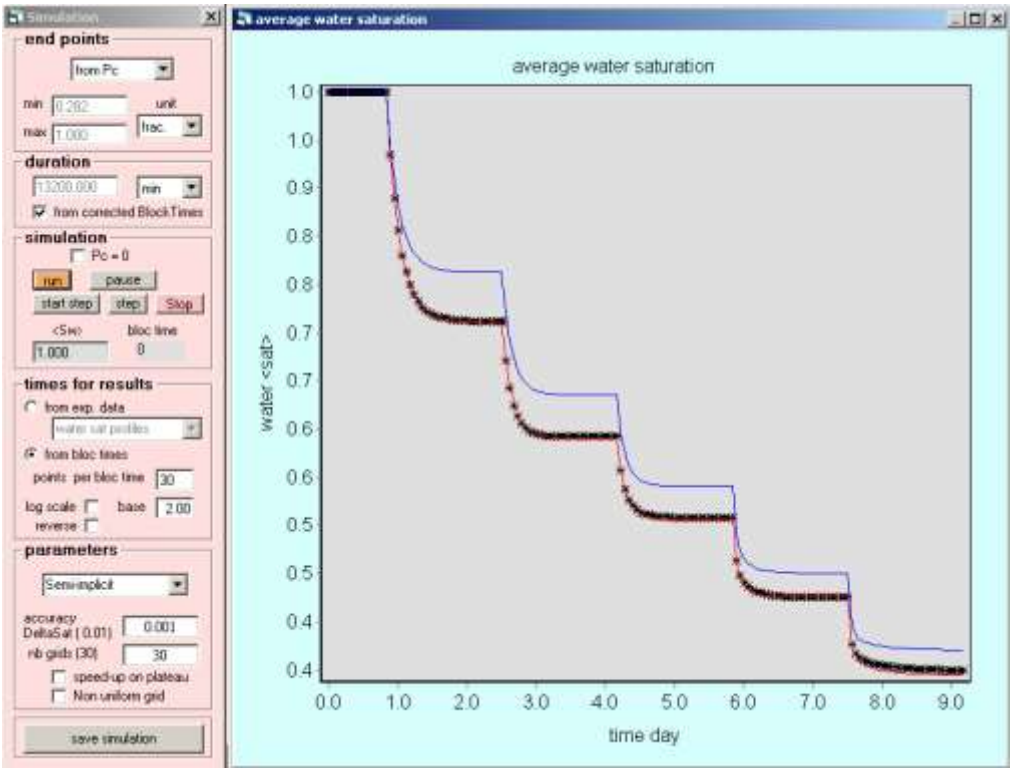


Figure 66: Simulation using the local Pc curve by both methods and comparison with the original data.

Usually users do not know the local Pc curve to control the quality of their calculation. This is where the quality control tool "**Pc(<S>) from Pc(S)**" is useful. Let's check the calculation of the average Pc(<S>) from the local Pc(S) curve obtained with the SCA 1991 Forbes method:

- file Tutorial_PcCentri_Drainage_Forbes_5_Pts.cyd;
- open the Pc centrifuge window (Figure 60) and choose the option "**Pc(<S>) from Pc(S)**";
- click "**calculate**";
- to display the calculated Pc(<S>) versus the experimental data, open the Pc(sat) graph and select curve "**edited points <sat>**" and "**calculated <sat>**".

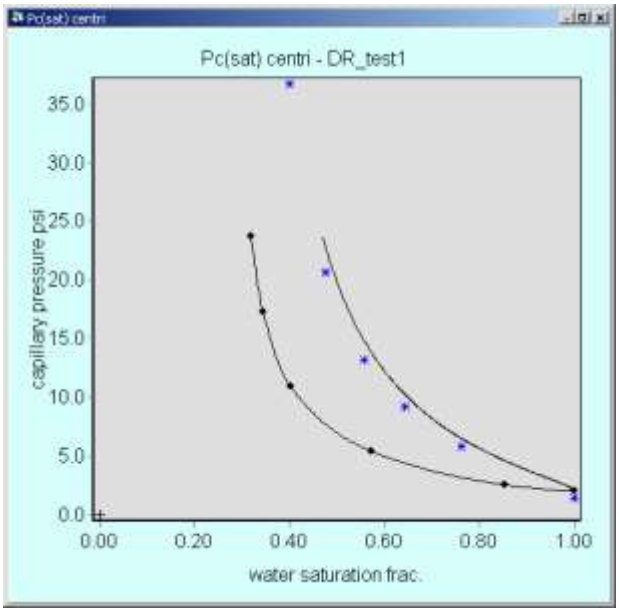


Figure 67: Comparison of the calculated and experimental Pc(<S>), calculation done on the Forbes SCA 1991 local Pc curve.

Figure 67 shows that the calculated Pc(<S>) from the local Pc obtained using the SCA 1991 Forbes method is not suitable. Production will be under estimate at each step confirming what simulation shown.

Relative Permeability – Hagoort

Details of the Hagoort's calculation can be find in its article "Oil Recovery by Gravity Drainage," SPE Journal, Vol.20, No.6, (June), pp.139-150, 1980. Basically, the Hagoort method is the analytical solution of the Buckley-Leverett equation for gravity drainage with the following assumptions:

- Capillary forces are negligible;
- The mobility of the non-wetting fluid is far more greater than the wetting one;
- The gravitational and centrifugal fields are equivalent.

These assumptions lead to very simple equations for the determination of the non-wetting relative permeability (Eq. 11 and 16 of Hagoort's article). Equation 16 gives directly the relative permeability versus time, the second one allows the saturation calculation versus time.

The CYDAR tool is simply the implementation of these equations. Calculations are done on fit curves. Although analytical calculation assumes a drainage displacement, CYDAR allows calculations on imbibition data as well; it is up to the user to evaluate calculation relevance.

Main steps are:

- Specifying block time (speed value);
- loading production data;
- Data fitting;
- Calculation.

Data fitting is a crucial step because:

- relative permeability is equal to the time derivative of the production data.
- CYDAR calculation starts at t=0, therefore analytical curve must start at t=0.

The "centri Kr" window, Hagoort part

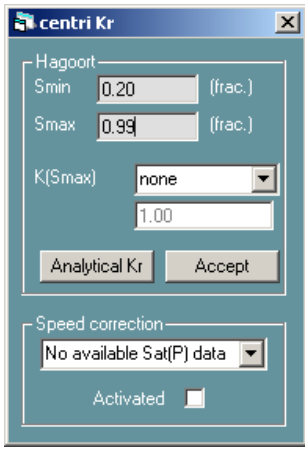


Figure 68: the "centri Kr" window, Hagoort part.

Click on the "**analytical Kr Hagoort**" button in the "**calculation**" frame of the main TPF window to open the window.

"Hagoort" frame:

- Smin and Smax are output. They give the calculated Kr's end points.
- K(Smax) in drainage or K(Smin) in imbibition fix the extremum values.
- "**analytical Kr**" launches calculation.
- "**accept**" button loads calculation to data points. Any existing Kr data are erased.

Remark: The K(Smax) (K(Smin) in imbibition) with option "none" means that only calculated values with saturation smaller or equal to one are kept. Saturation values bigger than one may occur due to fit values close to zero.

Corey fit

The calculated Kr curve can be fitted with the "**power**" fit function to obtain the Corey parameters ("**load/fit data**" frame, and select the corresponding Kr).

When displayed in log scale, the fit appears not optimal for lower values since the first point is taken as Kr=0 and corresponds to the asymptote (*Figure 69*). The saturation of the first point can be adjusted manually using the edit function (changing the X value), and pressing calculate. Figure 70 shows the result using a first point at S=0.17.

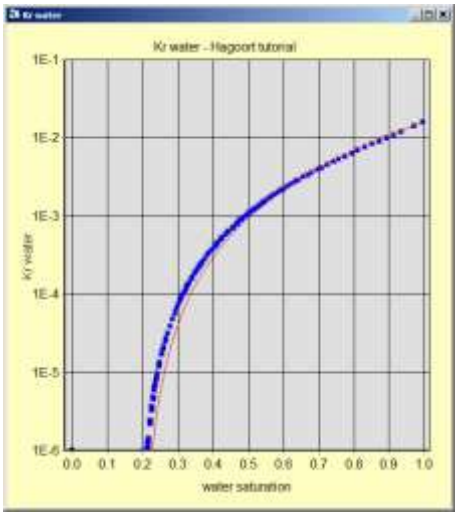


Figure 69: Corey fit of the tutorial, keeping the default value for the first point (S=0.2)

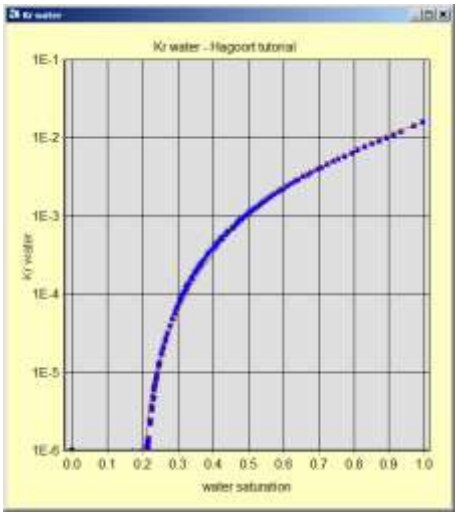


Figure 70 : Corey fit of the tutorial, adjusting the first point saturation at 0.17

Tutorial Hagoort

Files:

- tutorial_Hagoort.xls
- tutorial_Hagoort.cyd

This tutorial shows a one-step centrifuge drainage. The production data is a simulation done with:

- Pc=0;
- Corey relative permeabilities: Krw max=0.016 nrw=2.52, Krg=1 nrg=2;
- End points: Smin=0.2, Smax=1.

Starting a new project:

Common steps:

- Entering sample and flow properties
- Entering Experiment characteristics
- Entering Block times values
- Loading production data

First, edit the production data, here there are enough data points to use a simple linear interpolation. The user has to specify that fit starts at point (0,0) ("first point" to "value" Figure 71).

At this stage, the project is similar to tutorial_Hagoort.cyd.

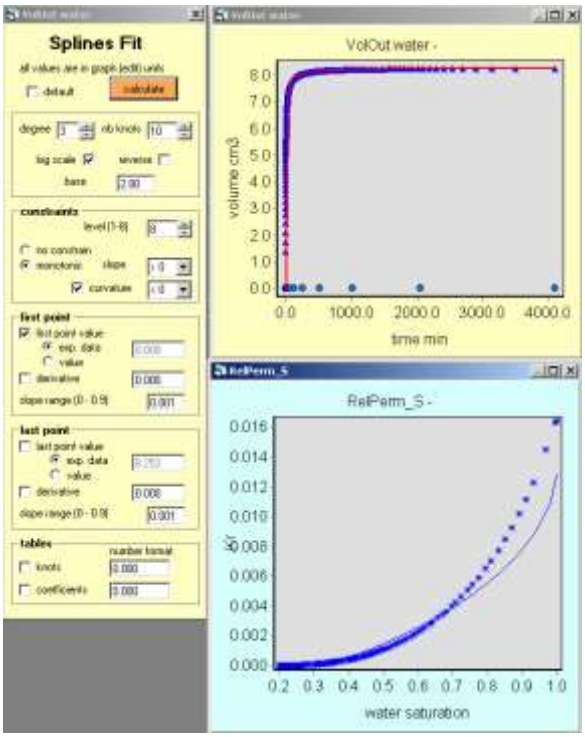


Figure 71: Hagoort calculation, effect of production data fit, splines with 10 knots.

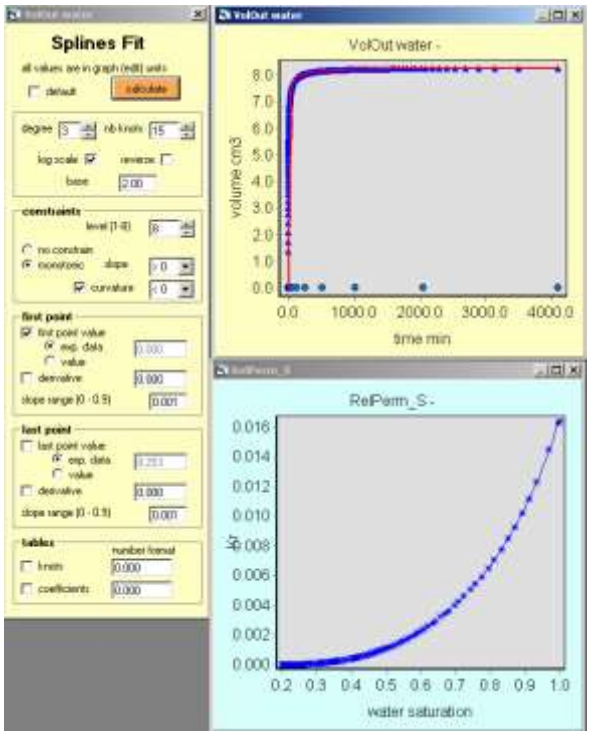


Figure 72: Hagoort calculation, effect of production data fit, splines with 15 knots.

Calculating Kr:

- Open the "**centri Kr**" window as explain above (Figure 68).
- Click on "**analytical Kr**": this launches calculation, a new "**analytical curve**" will be displayed in the "**Kr graph**".

  If the relative permeability is uniformly null, check all properties as densities, sample length, speed value in "**block time**"
- Click on "**accept**" button to load the calculation points to the data points, any existing data are erased.

Checking the result:

- The original relative permeability was $Kr_{w\,max} = 0.016$ and $n_{rw} = 2.52$;
- Open the "**data**" window;
- Select the data "**Kr water**";
- Select the "**power**" fit, click the "**Fit**" button;
- Fit calculation gives Kr max = 0.016 and n=2.87.

> **Useful Tip:** The result depends strongly on the editing step. For example, a fit done with "splines" will give different results depending on the number of knots (see Figure 71 and Figure 72).

In this particular case, one must ensure the constraints to be applied by choosing level 8. Otherwise, local variations may give non-monotonic relative permeability.

Multistep Kr

Speed correction

The average Pc curve calculated from the local Pc curve does not always fit the experimental points. Then the simulation, even with the real Kr, cannot fit the effluent production steps without Pc optimization. Trying to optimize Kr with inaccurate steps will lead to inaccurate Kr.

In CYDAR, the user has the possibility to correct speed values of block times in order to shift experimental points of the average Pc on the calculated curve.. This is an indirect method to correct for experimental uncertainties on speed values.

The principle is as follow: pressure values of the experimental data are shifted to fit the calculated point at the same saturation. Then the corrected values of speed are calculated from the corrected experimental average Pc curve.

Figure 73 shows an example of experimental average Pc curve shifted to fit the calculated curve. Experimental data points are red stars and the calculated curve is the black line.

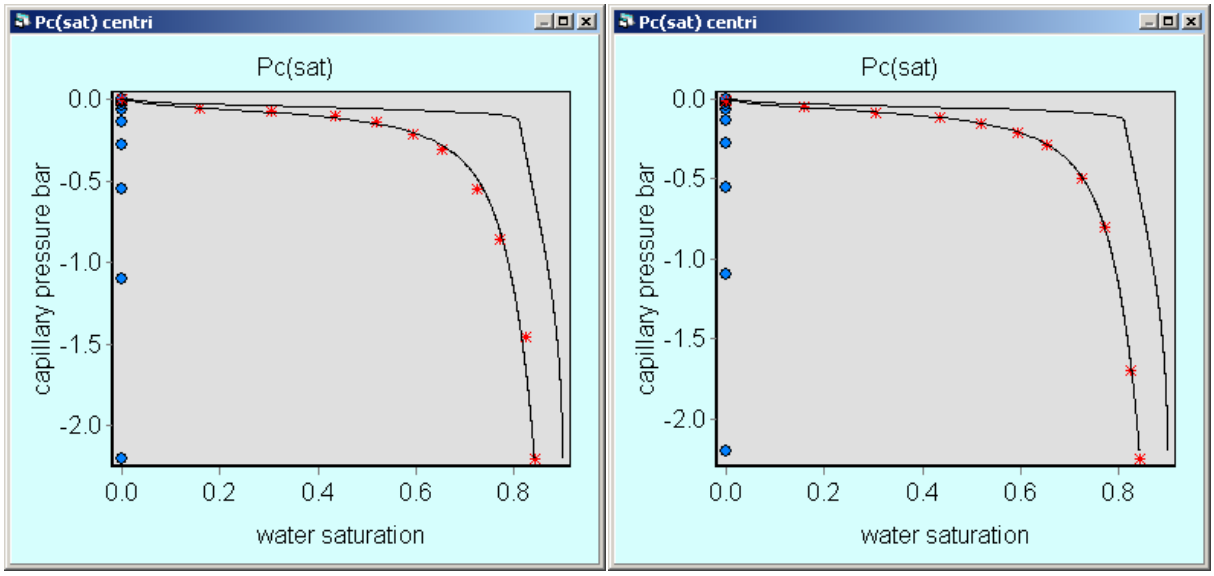
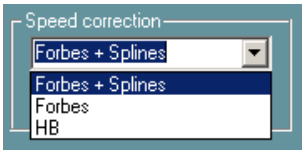


Figure 73: speed correction to fit calculated Pc(<S>).

To activate speed correction:

- perform Local Pc calculation (see § Local Pc from average)
- open the Kr centri window (Figure 68);
- choose which calculated average Pc curve to fit;



- check the box "**Activated**"

- The number of block times and the number of data points of "**sat(Pc)**" and "**sat(speed)**" must be identical.

The speed values in the block times table are then corrected. The experimental points on graph "**Pc(sat) centri**" (Figure 73 right) are updated. The "**Edit Points**" graphs of user input data "**sat(Pc)**" and "**sat(speed)**" will also display the corrected edit points.

To go back to non corrected values just uncheck the "Activated" box or reload the "**sat(Pc)**" or "**sat(speed)**" data.

Porous Plate

Porous plate is a direct measurement of the capillary pressure curve. At the inlet and the outlet a semi-permeable membrane to one of the fluids is placed. Then for each face, only one of the two fluids can flow trough, with their pressure fixed. At the equilibrium, pressures and saturations are uniformly distributed. This leads to the equality between the macroscopic properties measured at the two faces and the local ones. Then, the mean saturation is determined from the effluent volumes, and the capillary pressure directly from the difference between the fluid pressures.

Porous plate or membrane characteristics can be entered in the "**type of experiment**" window in the frame "**Porous plate**" (Figure 74). Three types of porous plate experiments are provided:

- "**no plate**": membranes have no flow resistance.
- "**one plate**": only the outlet membrane has flow resistance.
- "**two plates**": both inlet and outlet plates have flow resistance.

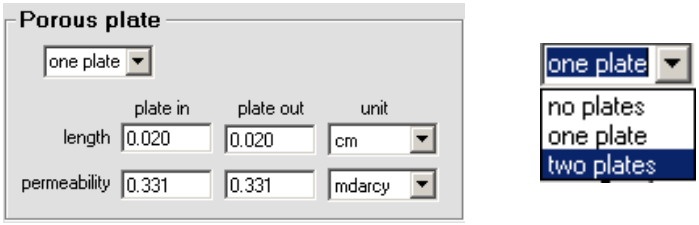


Figure 74: Porous plate characteristics.

"**One plate**" and "**two plates**" experiments are simulated as composite porous media. The sample is a block with properties given in the sample/fluids window as any homogeneous sample. Plates are blocks with length and permeability given in the "**type of experiment**" window (Figure 74). Table 5 shows specific membranes' relative permeabilities and saturation.

	Displaced fluid saturation	Displaced fluid relative permeability	Invading fluid relative permeability
Inlet	0	0	1
Outlet	1	1	0

Table 5: Membranes relative permeabilities and saturation in a porous plate experiment.

Because of the long times required to reach equilibrium, the design of the experiment is a very important task. CYDAR provides several tools for that purpose.

There are three ways to display capillary pressure curves:

- The common way, capillary pressure versus local reference fluid saturation. This is the capillary curve used in simulations. It is referenced as "**Pc**" in the "**Data points**" or "**curve display**" windows.
- The capillary pressure versus experimental saturation at end of step, "**end step sat**".
- The capillary pressure versus extrapolated saturation, "**extrapolated sat**". Asymptotic values of saturation are calculated using the multi-exponential fit (Figure 8).

These data can be loaded in the "**Data points**" and curves can be displayed on the same graph "**Pc porous plate**".

Heterogeneities

Heterogeneity data type

There are three types of heterogeneity in CYDAR. They are chosen in the "**Fluids and sample**" window with a combo box below the sample properties' input boxes.

Homogeneous

The sample has constant porosity and permeability. The user gives the length, the porosity, and the permeability values.

Composite

The sample is a succession of homogeneous blocks with given length, permeability and porosity.

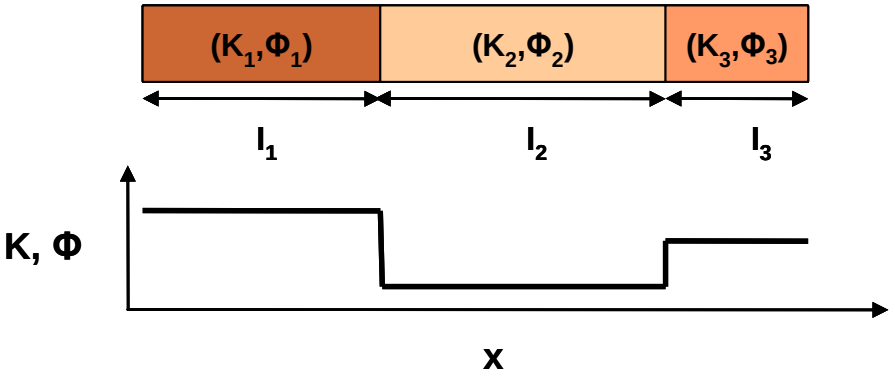
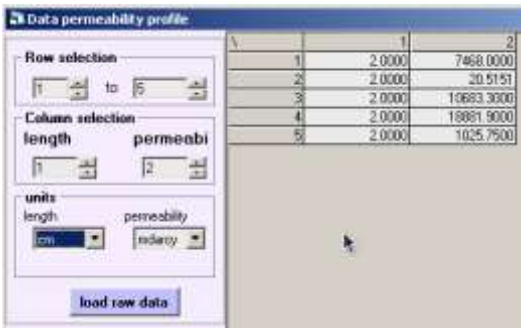


Figure 75: schematic composite sample.

First, the user loads data:



- Data are loaded in the "**data points**" window.
- Data are porosity and permeability values versus lengths of the blocks.
- Raw data are used as input in the simulation; any editing is useless and not possible. Permeability and porosity must have the same number of data points and the same length.

Next the heterogeneity type is selected in the "**Sample and fluids**" window (p. 41). The entries "**length**", "**permeability**," and "**porosity**" are then disabled and entry boxes show upscaled values: the total length, the harmonic mean for the permeability, and the arithmetic mean for the porosity.

Composite block lengths can be displayed on the porosity and the permeability graphs by selected "**Block length**" in graph edition panel (Figure 76).

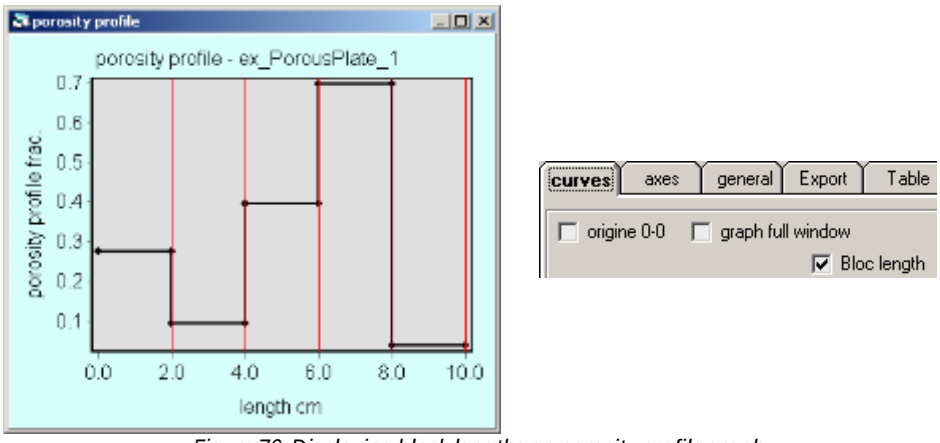


Figure 76: Displaying block lengths on porosity profile graph.

Profiles

The permeability and the porosity are continuous properties versus length.

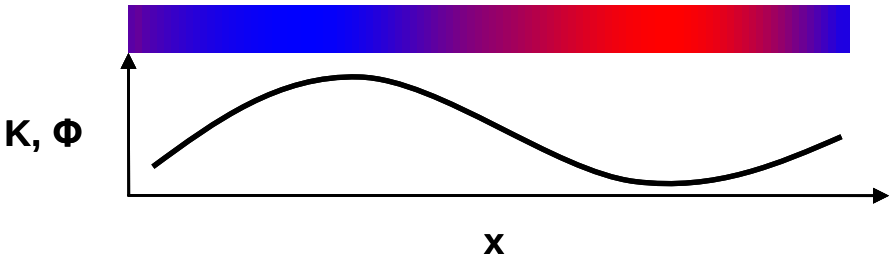
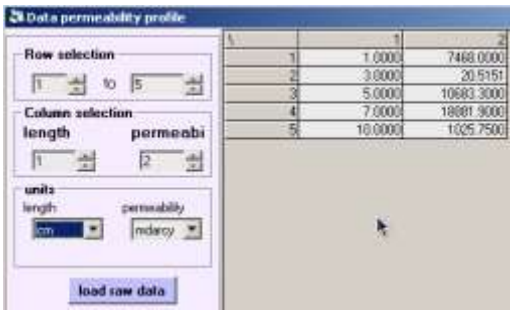


Figure 77: schematic sample characterized by porosity and permeability profiles.

First the user loads the data:



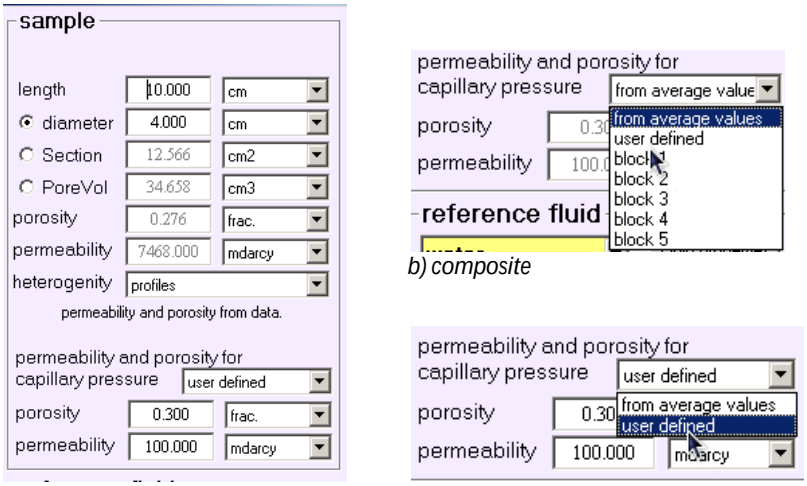
- Data are loaded in the "**data points**" window.
- Data are values versus length.
- The analytical fits are used as simulation input. The data can be edited as usual for moving, adding, or removing points.
- Without outlets values (x = 0 and x = L), automatic calculation is made assuming constant derivative. Therefore, it is recommended to check the data in the editing window.

When the heterogeneity type is selected in the "sample and fluids" window, the permeability and porosity text boxes are disabled. The length is a user entry because loaded data can not have values at the sample outlets. The entry boxes show upscaled values: the harmonic mean for the permeability and the arithmetic mean for the porosity.

Capillary pressure

In simulation the capillary pressure is calculated using the Leverett function. The user must indicate the porosity and permeability corresponding to the capillary curve used. Several options are possible (Figure 78):

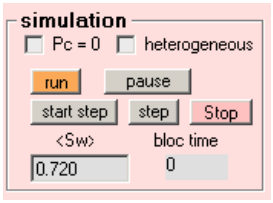
- "**from average values**": the harmonic mean for the permeability and the arithmetic mean for the porosity.
- "**user defined**": the permeability and the porosity are provided by the user.
- "**block**" number: for composite sample possibility to choose a given block.



a) *b) composite* *c) profile*

Figure 78: Sample properties: porosity and permeability input used for the Leverett calculation.

Simulation



If the sample heterogeneity is set to "**composite**" or "**profiles**" in the "sample and fluids" window, and if data are loaded, simulation can be performed with or without heterogeneities (check box "**heterogeneous**"). For homogeneous case upscaled permeability and porosity are used.

Composite

The mesh parameters apply on each block. For example, if the grids number is set to 10 for a composite with five blocks, this leads to a total of 50 grids.

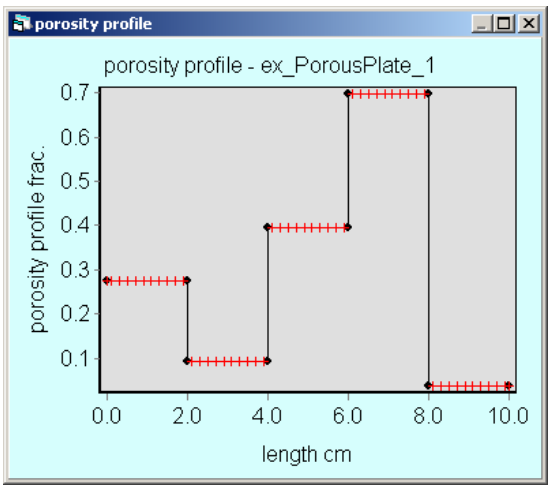


Figure 79: heterogeneity enabled and 10 grids per blocks.

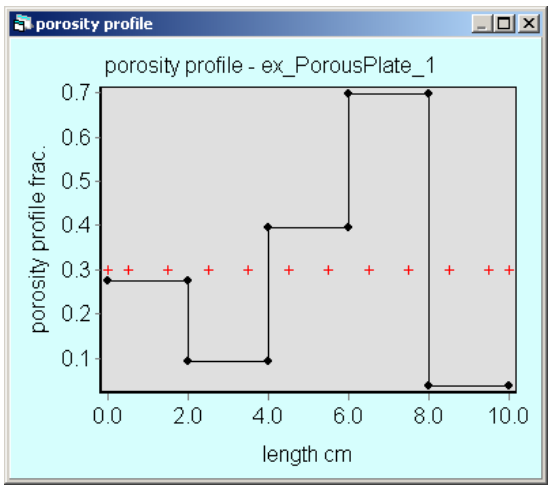


Figure 80: heterogeneity disabled and 10 grids.

Profiles

The mesh parameters are applied on the whole sample as for the homogeneous case.

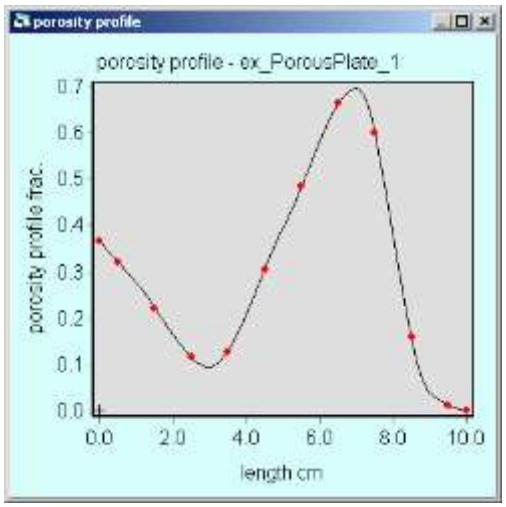


Figure 81: heterogeneity enabled and 10 grids.

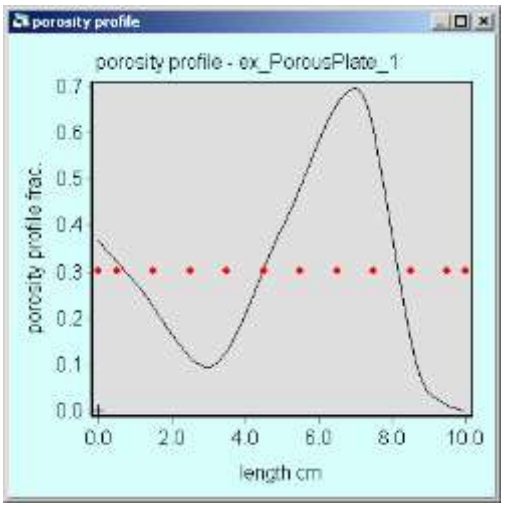


Figure 82: heterogeneity disabled and 10 grids.

Hysteresis

If drainage is performed until a uniform residual saturation S_{wi}, the following imbibition would follow a unique Pc curve defined between S_{wi} and a maximum. In that case only, one Pc curve is used for simulation.

Now let's assume that the first drainage is done until an intermediate uniform saturation S_{ini} (Figure 83). We want to simulate an imbibition from this state, the sample is at S_{ini} on the drainage Pc curve Pc^D, but we only have an imbibition Pc curve Pc^I defined from S_{wi}. How do we generate the imbibition curve defined between S_{ini} and its maximum end point?

The implementation in CYDAR is a work in progress. The model is based on several existing works see for example Kleppel et al. 1997 [1], Skjaeveland et al. 2000 [2] or chapter 12.6 in Aziz 1979 [3]. Its purpose is not at this stage to have a rigorous model but a tool to test qualitatively hysteresis effect.

Nomenclature and definitions used

- envelop curves: the main imbibition and drainage curves
- scanning curve: the intermediate curve
- P_c^I, P_c^D: imbibition and drainage envelop curves.
- P_c: scanning curve
- S_{ini}, S_{min}, S_{max}, S_{wi}: initial, minimum and maximum saturation, irreducible water saturation
- S_f: final saturation of the scanning curve
- Superscripts: I envelop imbibition curve, D envelop drainage curve

Pc scanning imbibition curve

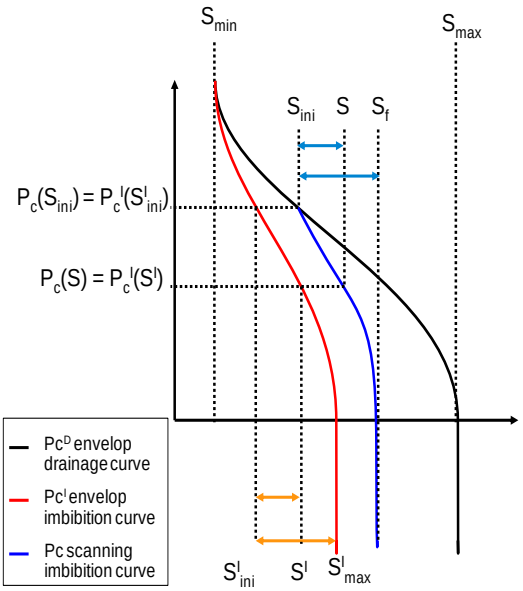


Figure 83: hysteresis, imbibition scanning curve generation.

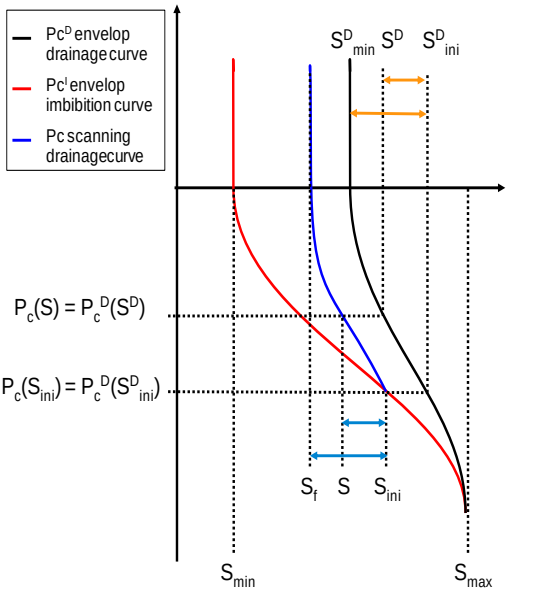


Figure 84: hysteresis, drainage scanning curve generation.

The principle is to generate a scanning imbibition curve Pc starting at S_{ini} from the envelop drainage curve Pc^D by calculating the saturation S of Pc such as $Pc(S)=Pc^I(S^I)$ (Figure 83). The model assumes that:

$$\frac{S^I - S^I_{ini}}{S^I_{max} - S^I_{ini}} = \frac{S - S_{ini}}{S_f - S_{ini}}$$

This gives the saturation on the envelop imbibition curve S^I. The final saturation of the scanning curve S_f is calculated from:

$$\frac{S_{ini} - S_{\min}}{S_{max} - S_{\min}} = \frac{S_f - S^I_{max}}{S_{max} - S^I_{max}}$$

Once S^I known, the scanning curve is simply given by $Pc(S)=Pc^I(S^I)$.

Pc scanning drainage curve

The principle is to generate a drainage scanning curve Pc starting at S_{ini} from the envelop imbibition curve Pc^I by calculating the saturation S of Pc such as $Pc(S)=Pc^D(S^D)$ (Figure 84) assuming:

$$\frac{S^D - S^D_{ini}}{S^D_{min} - S^D_{ini}} = \frac{S - S_{ini}}{S_f - S_{ini}}$$

this gives the saturation on the drainage envelop curve S^I. The final saturation of the scanning curve S_f is calculated from:

$$\frac{S_{\max} - S_{ini}}{S_{max} - S_{\min}} = \frac{S^D_{min} - S_f}{S^D_{min} - S_{min}}$$

Once S^D known, the scanning curve is simply given by $Pc(S)=Pc^D(S^D)$.

Loop

A hysteresis loop is simply a succession of imbibition-drainage, repeated several times. The implementation chosen here is not the popular model. The reason is essentially numerical. Let's recall it is still a work in progress.

Commonly the second scanning curve in a loop (light blue curve (Figure 85) meets the envelop curve at the origin of the first scanning curve (dark blue curve starting from black curve (Figure 85). For the moment in CYDAR the second scanning curve has the same final saturation calculation that if it is the first scanning curve.

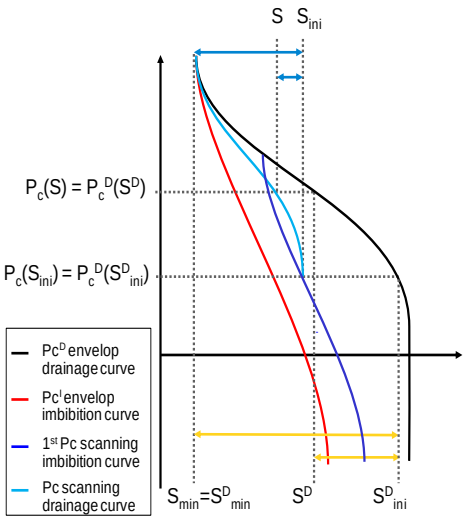


Figure 85 Hysteresis loop

Relative permeabilities

There is no specific treatment for the relative permeability curve unless the change of the local end points depending on the final saturation calculated for the Pc scanning curve.

Non-uniform initial saturation profile

In CYDAR, a non-uniform initial saturation profile can be used. The curve (saturation vs length) is loaded, edited, and fitted as usual (see § Load/fit data).

The type of profile is set in the "experiment type" window (see § "Type of experiment"). With hysteresis process the origin of the profile (drainage or imbibition experiment) must be known as seen above.

Inertial correction (version 2014)

The inertial correction is detailed in the "Permeability Module" chapter in the section "Inertial correction" page 35. In the two-phase flow module a local form of the Forchheimer equation is implemented for each fluid:

$$\left(F - \frac{dP}{dx}\right) = \frac{\mu}{K\,K_r}U + \beta\rho U^2$$

with β the inertial coefficient which has the dimension of the inverse of a length, *F* is the body force equal to ρg for gravity or $\rho\omega^2 r$ in centrifuge.

The inertial coefficients of both fluids are entered in the "fluid and sample properties window (Figure 34). By default the entries are hidden and can be displayed by checking the "show inertial parameter" box (Figure 86).

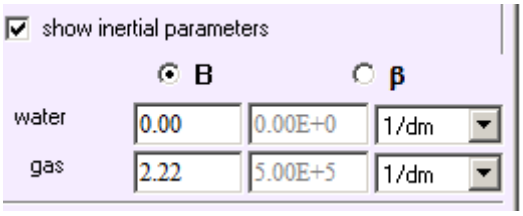


Figure 86: The inertial coefficient entries in the two-phase flow module.

The inertial coefficient can be entered as β or in its dimensionless form $B = \beta\phi\sqrt{K_d}$ (see page 35). According to the Forchheimer equation, there is inertial correction only if the fluid density and β are not equal to zero. Therefore if one fluid has a nonzero β but its density is zero, a warning is displayed next to the option button β, on the right.

The inertial correction is active in a simulation if the "inertial" checkbox is checked on the simulation window (Figure 47). This box is enabled if at least one fluid has nonzero values for its β and density. For example if β and the density are nonzero values for the non-reference fluid, the correction will be active in the simulation only for the non-reference fluid. If both fluids have both parameters not equal to zero then the correction will apply on the two fluids.

Electrical (version 2016)

Purpose

This module based on electrical measurements on local electrodes along the sample has two applications

- Determine the local saturation, assuming that the resistivity index is known
- Determine the resistivity index using the local saturation obtained by the numerical saturation and the measured effluent production

The two methods can be combined into an optimization loop

Principles

The water saturation may be estimated by the measurement of the resistivity index RI using empirical laws.

The resistivity index is the ratio between the resistivity at a given saturation R_S and the resistivity of the fully saturated sample R_0

$$RI(S) = \frac{R_S}{R_0}$$

Three models are implemented to calculate the water saturation from measurements:

- Archie's law

$$RI = S_w^{-n1}$$

- "bending down" and "bending up" laws from Fleury[16]:

$$RI = S_w^{-n_1}\frac{1+C}{1+CS_w^{-n_2}}$$

$$RI = S_w^{-n_1}\frac{1+CS_w^{-n_2}}{1+C}$$

C= 0 leads to the Archie's law

Experimental setup possibilities

The measurements are either voltage or resistance at different abscissa along the sample.

If the measures are resistances then the saturation at the electrodes location is calculated directly from the empirical law.

If the measures are voltages then the resistivity is calculated from the tension difference between two electrodes and the current.

Rload at the inlet

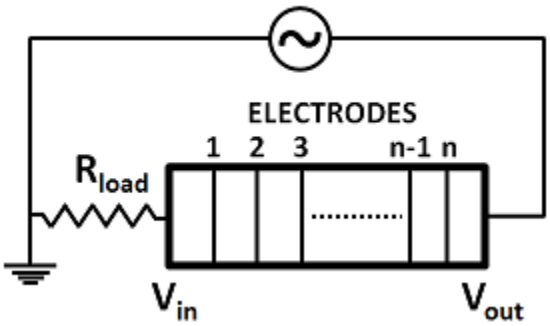


Figure 87: Rload at the inlet

The current is given by

$$I = \frac{V_{in}}{R_{load}}$$

Rload at the outlet

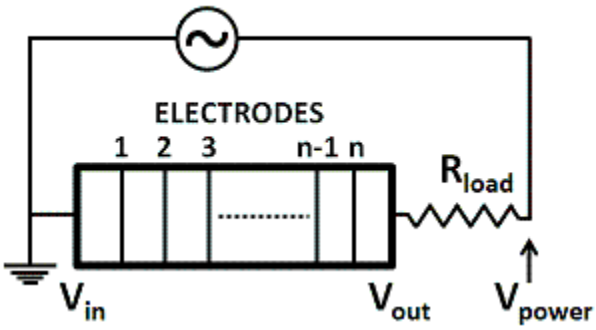


Figure 88: Rload at the outlet

The current is given by

$$I = \frac{V_{out} - V_{power}}{R_{load}}$$

Resistances calculation

The resistances are given by

$$R_1 = \frac{V_1 - V_{in}}{I}$$

$$R_i = \frac{V_i - V_{i-1}}{I}$$

$$R_{n+1} = \frac{V_{out} - V_n}{I}$$

The resistivity index along the sample are then calculated using the resistivity value when the sample is fully saturated

$$RI_i = \frac{R_i}{R_{0_i}}$$

Contact resistances

The two extreme resistances contain contact resistance. They may be corrected by using R_2 instead of R_1 and/or R_n instead of R_{n+1}.

Saturation calculation

Local saturations at the abscissa are calculated using one of the three empirical laws from the local resistivity indexes, for instance in the Archies' law case:

$$S_{w_i} = RI_i^{-\frac{1}{n1}}$$

The average saturation is calculated by:

$$< S_w >= \frac{\sum dx_i \, \Phi_i \, A \, S_i}{V_p}$$

the dx_i are the distances between electrodes if the measures are voltage, around each electrode if the measures are resistances. Φ_i is the porosity at the middle of dx_i. A is the section and V_p is the total pore volume.

CYDAR module

The sample characteristic, the abscissa, the profiles and the saturated profile are compulsory. If the profiles are voltages, they must contain V_{in} and V_{out}.

V_{in} and V_{out} are not counted in the data with the electrode abscissa.

The experimental setup parameters are set in the window "Electrical window" Figure 89 and the data versus times and the profiles are loaded in the "Data Points" windows.

Electrical method: setup window

Entries of the window are fairly obvious, see below for some explanation.

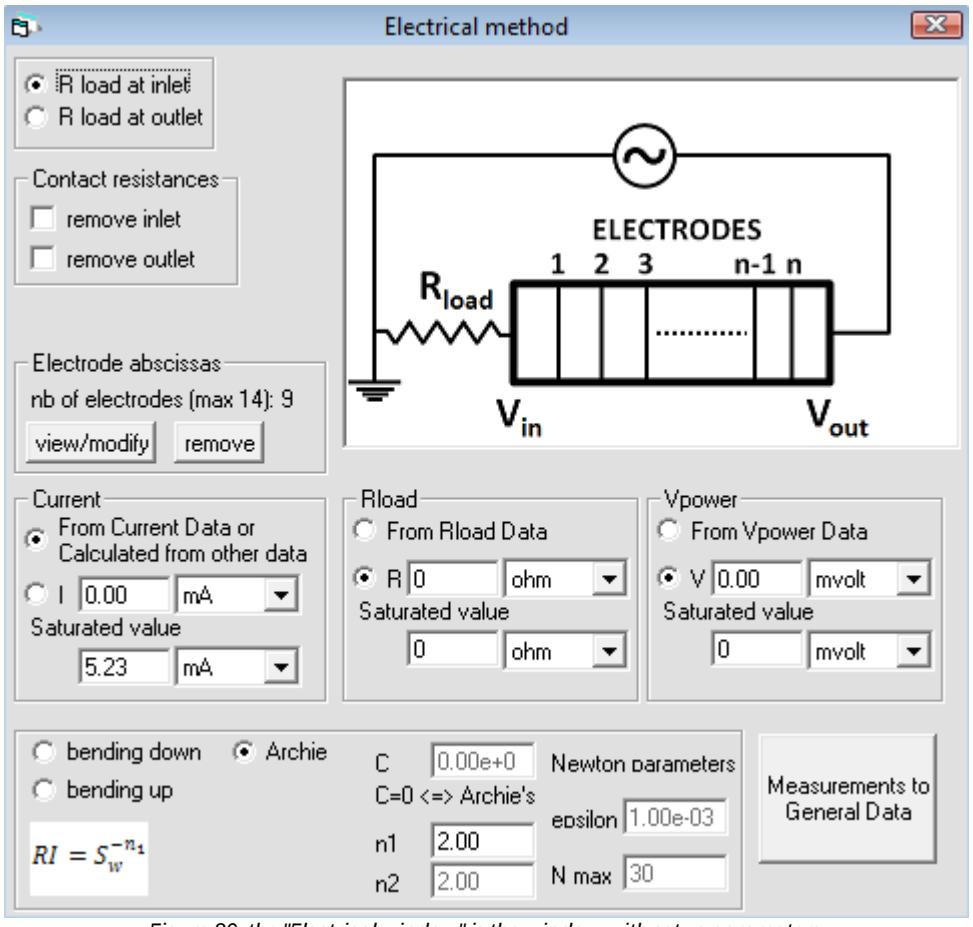


Figure 89: the "Electrical window" is the window with setup parameters

The experimental data are resistances

The only parameters used in that case in Figure 89 are the empirical law parameters at the bottom of the window.

The only data needed are the profiles and the values along the sample corresponding to the fully saturated case.

The data are loaded in from the common data points window see below for some details.

The experimental data are voltages

This case may leads to different combination of parameters and data inputs.

The current is directly entered

The current is either an input as a data versus time or a constant value. In that case Rload and Vpower are useless.

If it is a data versus time then the current value corresponding sample is compulsory.

If a value is given for the value of the saturated sample then this will be the value used even if the current is set to a constant value.

The current is calculated from other data

The current is calculated according to Rload, V_{in}, V_{out} and/or V_{power} depending on the setup.

If Rload is at the inlet then V_{power} is not needed:

- The Rload may be either a constant or a data versus time loaded in the data points window. If Rload is not set to a constant value, the value corresponding to the saturated case is compulsory.

If Rload is at the outlet:

- in addition to Rload, V_{power} is needed and may be entered as a constant or a data versus time. In the later case the value of the saturated sample is compulsory.

Saturation calculation

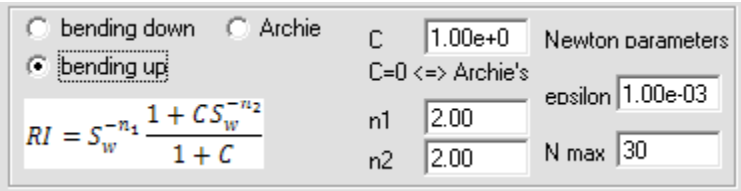


Figure 90: frame with the parameters used for saturation calculation

In the case of the Archie's law this is straight forward.

For the two other laws, the calculation is performed using a Newton-Raphson iteration scheme. The frame shown Figure 90 contains the different parameters. epsilon is the stopping criterion and Nmax is the maximum of iteration.

Input data

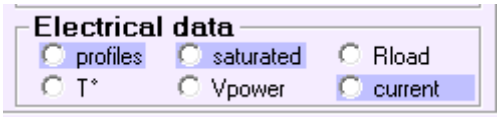


Figure 91: data related to electrical method in data points window

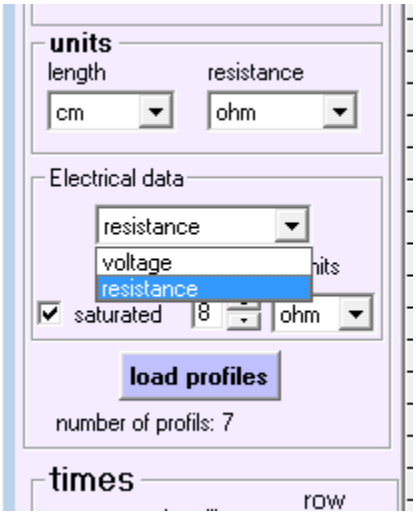


Figure 92: Ascci window to load profiles with the possibility to choose the type of data voltage or resistance.

- "profiles" are the measurements along the sample at different time. They are loaded as other profiles in CYDAR. They can be either voltages or resistance measures. The choice is made in the Ascii window (see Figure 92). The values along the sample for the saturated case may be entered the profiles by giving the column corresponding to it (see).
- "saturated" the measurements along the sample done with a fully saturated sample, can be either voltage or resistance.
- "Rload": Rload values versus time.
- "Current": data versus time. If exist and the option from data is chosen then this is this data which is used.
- "Vpower": data versus time.
- "T°": temperature data versus time for future implementation of temperature correction.

Remark: The times for the different data must be identical.

Output data

When all needed parameters and data are present the saturation versus time S(t) at the different electrode abscissa and the average saturation versus time <Sw> calculated from the measurements may be displayed on different graphs using the "graphs and tables" window.

The "classical" CYDAR saturation profiles and average saturation entered via the data points window are not automatically calculated. They must be explicitly loaded from the values calculated from the measurements by pressing the button "measurements to general data" at the bottom right of the "electrical window" Figure 89.

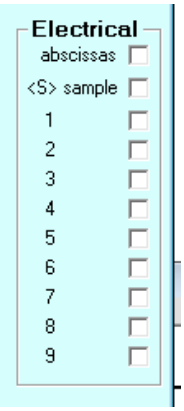


Figure 93: Graphs relative to electrical data

TPF Reporting

Principle of ASCII file

The report can be described as two parts: one with parameters, sample properties (Figure 94); and the second part with the data.

On Figure 94:

Block 1:
- General information: date, client name, operator, etc.

Block 2:
- Sample data: name, size, heterogeneity type, porosity, permeability.
- Experiment data: type, initial saturation, external pressure, gravity, etc.
- Simulation parameters: accuracy, mesh parameters, etc.

Block 3:
- Fluids properties: viscosity, density, compressibility, etc.

Block 4:

- Pc and Kr analytical parameters

The second part summarized data and block time values:

- Analytical data of permeability and porosity profiles used in simulation
- Block times values: incremental, cumulative time, experimental type inputs (pressures, flow rates, speed) and columns with pressure and volume values allowing easy block times drawing on graphs using template.
- "Raw Data", "Edit Points", "Analytical", "Simulation", "Simulation BackUp" for the main data: capillary pressure, relative permeabilities, delta pressure, outlet volumes, average saturation, volume balance.
- With additional data for specific cases as, for example, in centrifuge experiment: Pc(<S>); forbes + splines if calculation have been done.

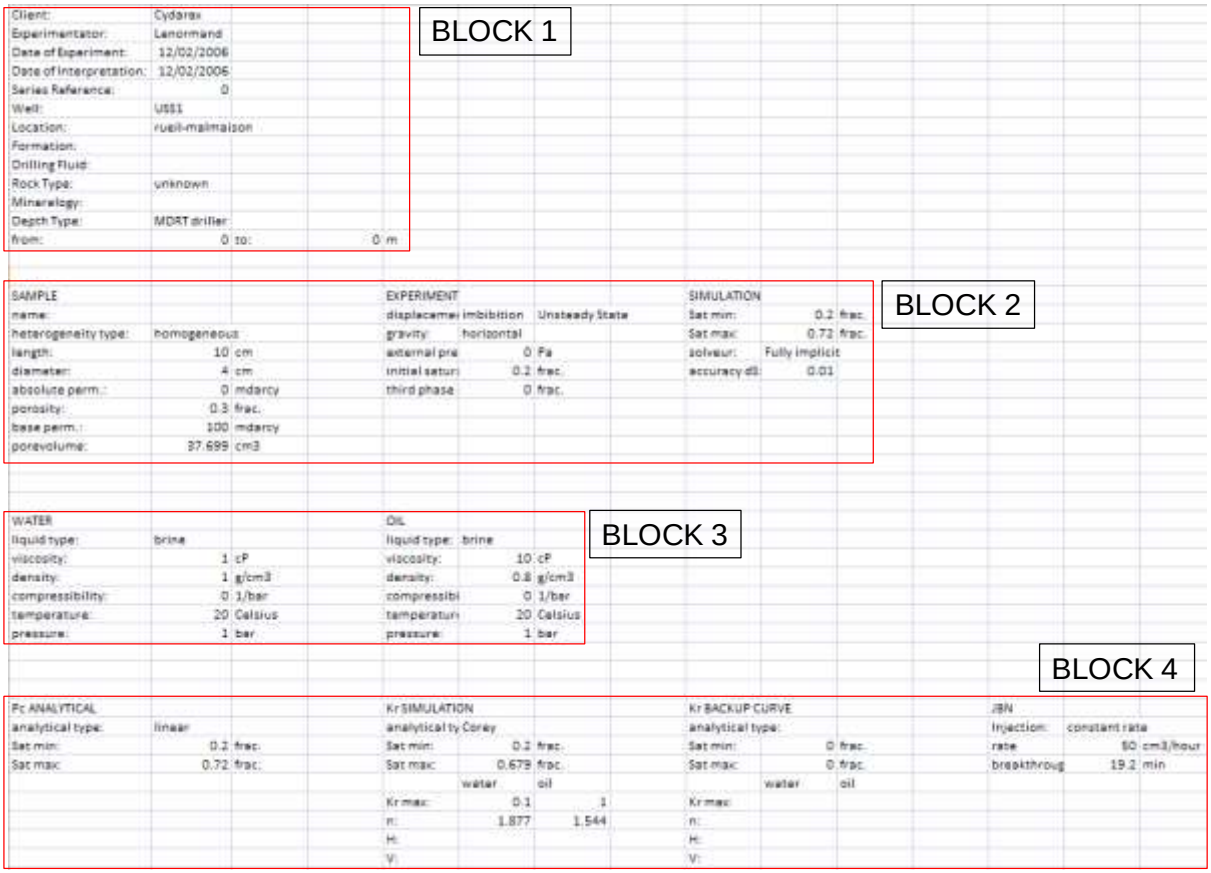


Figure 94: First part of an ASCII report for an unsteady-state experiment.

Using template:

The main goal of the ASCII report is to be easily imported in spreadsheet programs like OpenOffice or Microsoft Excel. Complex layouts (with graphs, colors, company logo) can then be created using templates. Figure 95 shows a template example.

When using Templates with TPF, relative permeabilities should be plotted with 30 points, and time unit in Block Times should be the same as graph units.

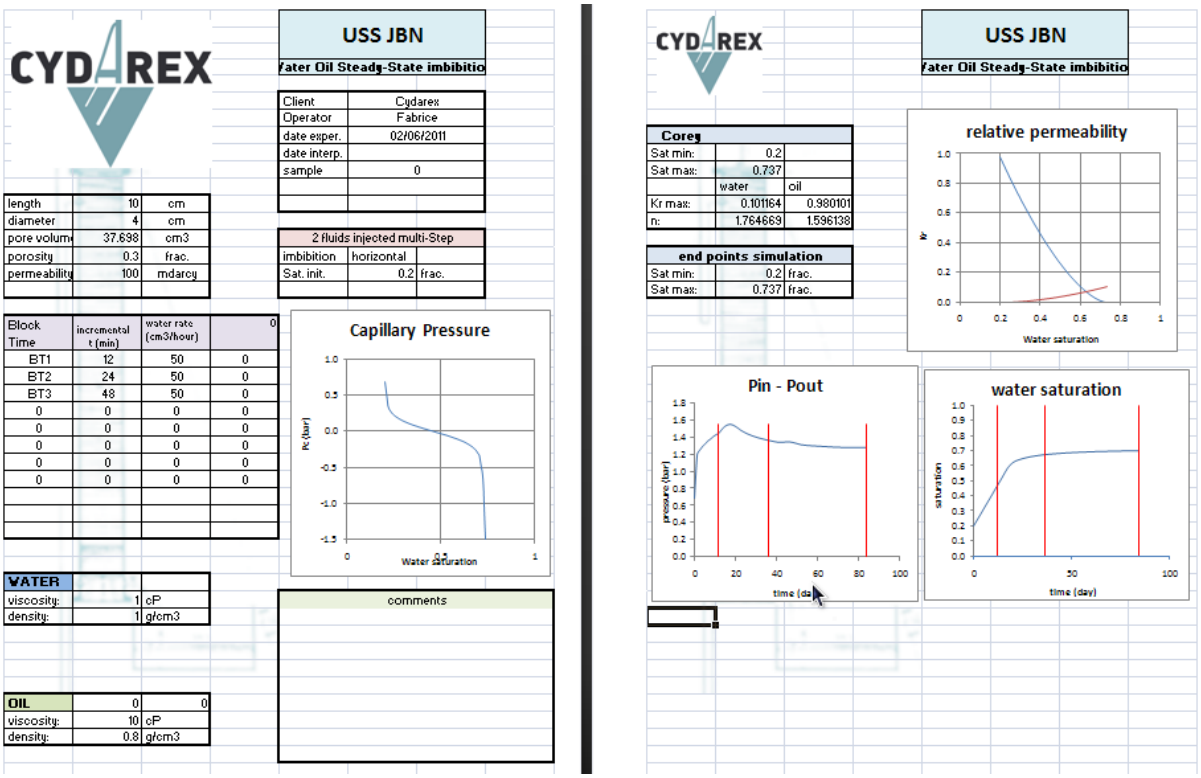


Figure 95: example of a more complex layout using template.

Curve Fitting Tool

The Curve Fitting tool, accessed through CYDAR main window (pg. 26), is an independent module that gives access to CYDAR fitting functions, data manipulation, and allows several calculations.

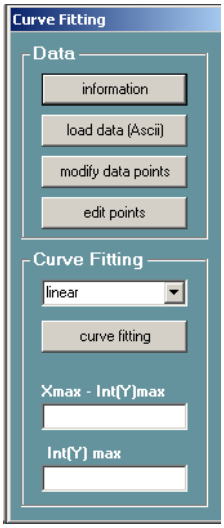


Figure 96: Curve Fitting tools main window.

"**information**" opens the information window, common to all modules (sample, operator, client …).

"**Load Data (Ascii)**" opens a window for data loading, editing, or fitting (page 45).

"**modify data points**" allows modification of data.

"**edit points**" allows data editing (pg. 16).

Once data are entered, they need to be fitted with the "**curve fitting**" tool, describe on pg. 17.

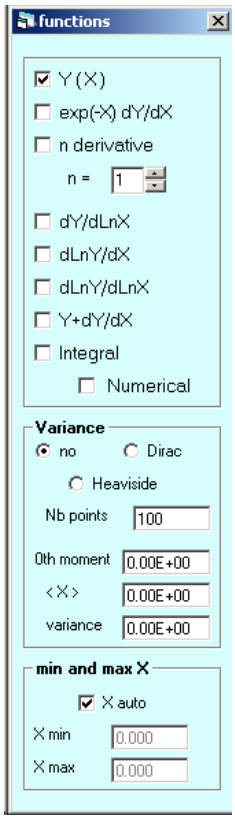


Figure 97: View window.

Once data are entered and fitted, the "**View"** menu gives access to additional graphs.

Appendices

Numerical methods

Depending on the modules, numerical calculations are based either on explicit or implicit methods.

Explicit methods calculate the state of a system at a later time from the state of the system at the current time, while implicit methods find a solution by solving an equation involving both the current state of the system and the later one.

Mathematically, if $Y(t)$ is the current system state and we want to find $Y(t + \Delta t)$ at the later time, t+ Δt , (Δt is a small time step), then, for an explicit method:

$$Y(t + \Delta t) = F(Y(t))$$

while for an implicit method one solves an equation:

$$G(Y(t), Y(t + \Delta t)) = 0$$

Newton-Raphson and Euler are two examples of powerful methods used to solve implicit schemes in CYDAR.

Newton-Raphson method:

Flow equation can be written as $\mathbf{F}(t) = 0$ and then the system can be solved with the Newton-Raphson iterative method.

In one dimension, the Newton-Raphson Algorithm to solve $f(x) = 0$ is:

- First guess: $x_0,\ k = 0$
- Newton-Raphson loop until $\|x_{k+1} - x_k\| < \varepsilon$:
- Solve $\frac{\partial f(x_k)}{\partial x}\ \Delta x = -f(x_k)$
- $x_{k+1} = x_k + \Delta x$
- $k := k + 1$

The solution is the last x_{k+1}.

Remark:

It exists different convergence criteria like the relative error $\|x_{k+1} - x_k\| < \varepsilon\ x_k$ or on the residual as follow $f(x_k) = \varepsilon\ f(x_0)$. The choice of the difference, $\|x_{k+1} - x_k\| < \varepsilon$, as exit test is based on empirical work done over several boundary, initial and experimental conditions.

Euler implicit:

The Euler implicit method is also known as the backward Euler method. Let's consider the following equation:

$$\frac{\partial y(t)}{\partial t} = f(y(t))$$

The purpose is then to use the finite difference:

$$\frac{\partial y}{\partial t} = \frac{(y^{n+1} - y^n)}{\delta t}$$

and to solve iteratively, with a Newton-Raphson method for instance:

$$y^{n+1} = y^n + \delta t\ f(y^{n+1})$$

Transient permeability module:

The flow equation is $\frac{\partial \varphi\, \rho}{\partial t} + \nabla(\rho\, \mathbf{U}) = 0$ and can be written in term of pressure in one dimension as follow on each grid:

$$\varphi(P)\, c(P) \frac{\partial P}{\partial t} + \frac{\partial \varphi(P)}{\partial t} + U(P)\, c(P) \frac{\partial P}{\partial x} + \frac{\partial U(P)}{\partial x} = 0 \quad \Leftrightarrow \quad f(t,x) = 0$$

Because this is solved on each grid, we have a non linear system to solve $\underline{F} = 0$ Then *P* and *F* are vector of length n (n being the number of grids): $\underline{P} = (P_1 \quad \cdots \quad P_i \quad \cdots \quad P_n)$ and $\underline{F} = (F_1 \quad \cdots \quad F_i \quad \cdots \quad F_n)$. The Newton-Raphson algorithm is the same as above with the following system to solve iteratively:

$$\underline{P}^{k+1} = \underline{P}^k - \underline{\underline{J}}^{-1^k}\, \underline{F}^k$$

With $\underline{\underline{J}}^{-1}$ the inverse of the Jacobian matrix of $\underline{F}$ in accordance with $\underline{P}$.

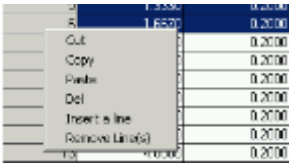
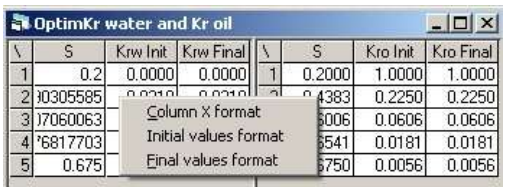
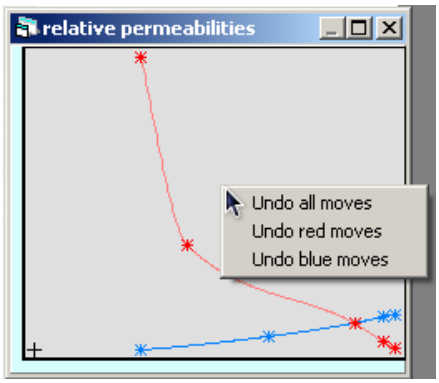
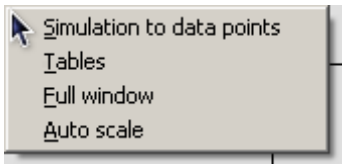
Two-phase flow module:

CYDAR has two fully implicit solvers implemented: uncompressible and compressible.

For Two-Phase Flow simulations, "Fully implicit" means that the saturation and the pressure are solved both implicitly.

The well-known "IMPES" method (Implicit Pressure Explicitly Saturation) has been removed in the recent version of CYDAR, due to its lack of accuracy.

Right click menus

WHERE	ACTION	
Input tables Spreadsheet	removing, inserting lines	
Optimization per points: Kr and Pc tables.	Formatting numbers by column	
Optimization per points: Kr and Pc graphs.	Undoing all moves since the beginning of the optimization.	
XY Graphs	Loading simulation curve as data points (not available for graphs with several variables displayed like “Pc centri”, Kr, etc.). Opening data tables. Activating Full window display Axes auto scale	

Shortcuts

SHORTCUTS	WHERE	ACTION
Ctrl+F	Text boxes Input tables and spreadsheet	Formatting a number or a column in a table.
Ctrl+Z	Optimization per points: Kr and Pc tables and graphs.	undo the last move only, if no other action has been done
Ctrl+Z	Data editing	five steps undo stack
Ctrl+Y	Optimization per points: Kr and Pc tables and graphs.	Redo the last undo, if no other action has been done
Ctrl+Y	Data editing	five steps undo stack
Ctrl+w	Any CYDAR's windows	Copy to the clipboard
Ctrl + click on 	Graph Windows	Close all graph windows
Shift + click	On data point in Graph	Shows a window with X-Y values of point

Troubleshooting

Here is an overview of know issues and solutions.

Error message during installation

Depending on the operating system, some messages might occur during installation. Choose the option "ignore" and continue. When prompted for files "not newer than the existing one," use the option to keep the existing file as recommended.

The installation software cannot be launch

If Windows is set to use a Latin alphabet, but the installation software path contents letters from a different alphabet (Hebrew, Arabic, Cyrillic), the installation program might not be able to access the installation files.

If the installation software cannot be launch, it is recommended to use only Latin letters in its path.

If the user name contents non-Latin letters, you can create a temporary user, and run the installation software from the temporary user. Once installed, CYDAR™ should run without problems even if non-Latin letters are present.

Installation software gets stuck

Under Windows 8 or Windows 10, the installation software gets stuck and doesn't finish the installation

If the installation software freezes and doesn't finish the installation, we recommend to attempt installation with an Internet connection, as Windows 8 or 10 might be installing additional components needed for the installation.

Also, if the installation blocks with the following message: "Microsoft data access components 2.0 (not responding)", kill the process when prompted and let the installation finished. The installation should be fine.

Text on CYDAR™ windows is impossible to read

It may happen that CYDAR™ looks as shown on Figure 98.

This problem is due to the fact that the font in each object is too large compare to the object, and the problem can have several origins:

1. In Windows settings, in "Display properties/Settings/Advanced", make sure that the DPI setting is set at a minimum of 96 DPI. CYDAR™ is not designed to run with a DPI below 96.
2. Under some circumstances, Windows uses a different font for MS sans serif, and this font is slightly larger. The original font can be reset using the registry editor:
   click on the Windows logo at the bottom left of the screen (formerly the "Start" button), and in the tab "Search for programs and files", type: "regedit". This opens the registry editor.
   In the registry editor, navigate to: HKEY_LOCAL_MACHINE\SOFTWARE\Microsoft\Windows NT\CurrentVersion\Fonts\MS Sans Serif 8,10,12,14,18,24.
   By double-clicking on the font MS Sans Serif, the value data should read: "SSERIFF.FON"
   If it does, change it to: "SSERIFE.FON" – Note that there is one letter change: a F becoming a E.
   Press okay, and close the Registry Editor. You need to restart your computer for the change to take effect.

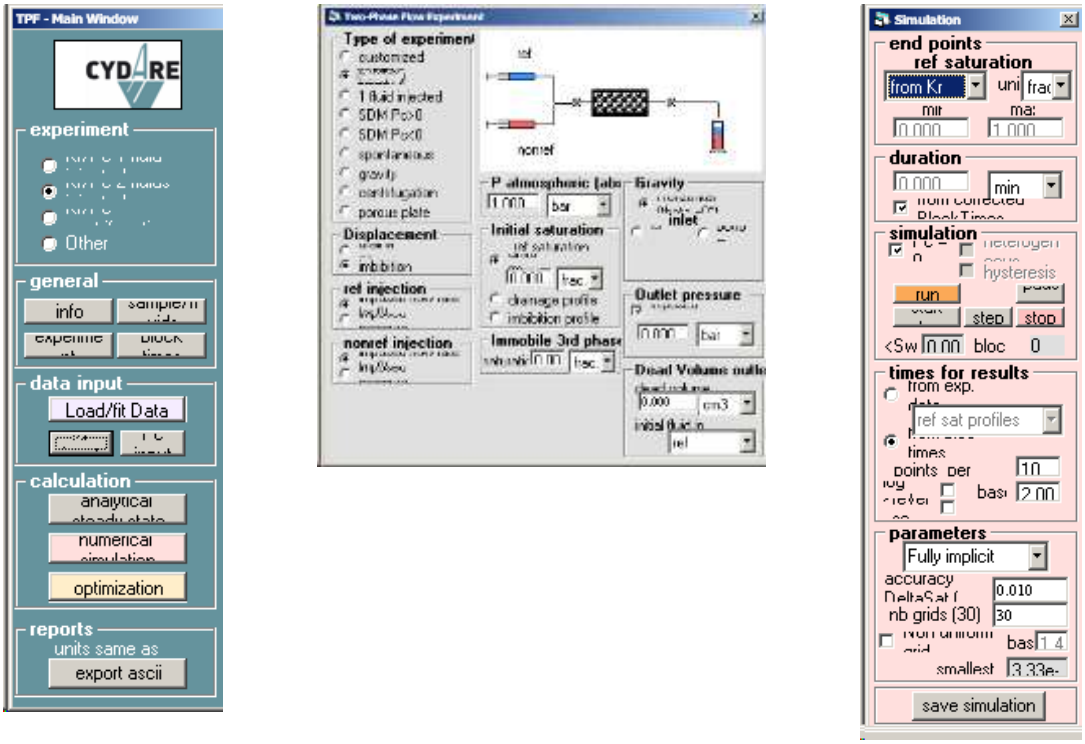


Figure 98: Problem in font size.

Issues when copy-pasting with Excel

When copy-pasting between CYDAR and Excel, numbers are not transferred properly. This is usually due to a mismatch between CYDAR number separator (the comma or dot used to denote decimals) and Excel's.

CYDAR uses the number separator set by the operating system. CYDAR can work with either comma or dot. Problems can occur when Excel is set to a different separator than the operating system.

To solve this problem, open "Excel options/Advanced" in Excel, and make sure that "Use system separators" is checked (Figure 99).

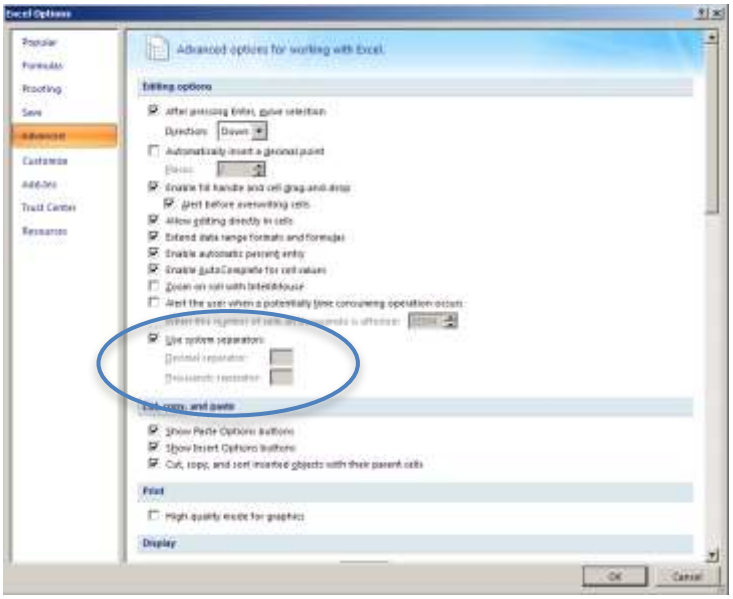


Figure 99: Excel option to set decimal separator.

References

1. J. Kleppe, P. Delaplace, R. Lenormand, G. Hamon, E. Chaput, "Representation of Capillary Pressure Hysteresis in Reservoir Simulation", SPE 38899, SPE Annual Technical Conference and Exhibition held in San Antonio, U.S.A., 05-08 October, 1997.
2. Skjaeveland S.M., Siqveland L.M., Kjosavik A., Hammervold Thomas W.L., Virnovsky G.A., "Capillary Pressure Correlation for Mixed-Wet Reservoirs", SPE Reservoir Eval. & Eng. 3 (1), February 2000.
3. Aziz K., Settari A., Petroleum Reservoir Simulation, 1979, Applied Science Publishers.
4. Lomeland F., Ebeltoft E. and Thomas H. W, "A new versatile relative permeability correlation", *Reviewed Proceedings of the 2005 International Symposium of the SCA, Abu Dhabi, United Arab Emirates, October 31 - November 2, 2005, SCA 2005-32*.
5. Amott, E., "Observations relating to the wettability of porous rock", *Transactions of the AIME* (219), 156–162, 1959.
6. Leverett, M.C. "Capillary behaviour in porous solids". *Transactions of the AIME* (142): 159–172, 1941.
7. Klinkenberg, L. (1941). "The permeability of porous media to liquids and gases." *Drilling and Production Practice*: 200-213.
8. Purcell, W. R. (1949). "Capillary pressures, their measurement using mercury and the calculation of permeability therefrom." *Trans AIME* (February): 39-48.
9. Johnson, E. F., D. P. Bossler and V.O. Naumann (1959). "Calculation of relative permeability from displacement experiments." *Journal of Petroleum Technology* (January): 61-63.
10. Jones, S. C. and W. O. Roszelle (1978). "Graphical techniques for determining relative permeability from displacement experiments." *Journal of petroleum technology* (may): 807-817.
11. Swanson, B.F., "A simple correlation between permeabilities and mercury capilary pressures", *Journal of Petroleum Technology, December, 2498-2504, 1981.*
12. Dullien, F. A. L., "Porous media. Porous structure and fluid transport", *San Diego, Academic Press Limited, inc.*, 1992.
13. Thomeer, J.H., "Air permeability as a function of three pore-network parameters", *Journal of Petroleum Technology*, 1983.
14. Rapoport, L. A. and Leas, W. J.: 1953, Properties of linear waterfloods, *Trans. AIME* **216**, 139–148.
15. Jones, F.O. and Owens, W.W.: A laboratory study of low-permeability gas sand, Journal of Petroleum Technology, Sept. 1980, 1631-1640.
16. Fleury, M.: Advances in resistivity measurements using the frim method at reservoir condition, application to carbonates, SCA2003-31, 2003.