



Fig. 5.
Appareil destiné à déterminer la loi
de l'écoulement de l'eau à travers le sable.



CYDAR-SCAL

User-manual

CYDAR-SCAL USER MANUAL

CYDAR-SCAL USER MANUAL	2
CYDAR-SCAL OVERVIEW	6
PC AND PORE SIZE DISTRIBUTION (MICP).....	6
ABSOLUTE PERMEABILITY.....	7
DISPERSION (CYDAR 2017)	7
TWO-PHASE FLOW EXPERIMENT	8
RELATIVE PERMEABILITY	9
CENTRIFUGE CAPILLARY PRESSURE.....	9
CYDAR MAIN FEATURES	10
<i>Windows environment</i>	10
<i>Data input</i>	10
<i>Numerical calculation</i>	10
<i>Exporting results</i>	10
<i>Data smoothing</i>	10
<i>Graphical display</i>	11
<i>Reporting</i>	11
EXPORTING RESULTS	12
<i>Graph</i>	12
<i>Simulation and analytical data</i>	12
<i>Reporting</i>	12
<i>Copying a graph or a frame</i>	12
NEW PROJECT WINDOW.....	13
PC & PORE SIZE DISTRIBUTION MODULE	14
MAIN WINDOW	14
LOADING EXPERIMENTAL DATA.....	15
TYPES OF CURVES	15
PORE SIZE DISTRIBUTION	16
PERMEABILITY CALCULATION	16
CAPILLARY PRESSURE	18
<i>J Leverett Function</i>	18
<i>Transition zones</i>	18
<i>Water cut</i>	19
DISPLAYING GRAPHS	19
TUTORIAL FILES.....	19
<i>Tutorial_Hg_SampleA</i>	19
<i>Tutorial_Hg_Brauvillier</i>	19
PERMEABILITY MODULE	20
PERMEABILITY MAIN WINDOW	20
SAMPLE AND FLUID PROPERTIES.....	20
KLINKENBERG CORRELATIONS	21
STEADY-STATE EXPERIMENT	21
<i>Data points steady-state</i>	21
<i>Calculate permeability: steady-state</i>	22
<i>Inertial correction</i>	23
<i>Klinkenberg correction</i>	24
<i>Tubing correction</i>	24

TRANSIENT EXPERIMENT	24
<i>Data points transient / unsteady state</i>	24
<i>Permeability pulse decay</i>	25
<i>Parameter menu transient:</i>	26
TUTORIAL FILES	26
<i>Tutorial_Perm_Klinkenberg</i>	26
<i>Tutorial_Perm_Inertial</i>	27
<i>Tutorial_PERM_Pulse_decay_Analytical</i>	27
DISPERSION MODULE	28
DISPERSION MAIN WINDOW	28
SAMPLE AND EXPERIMENT PROPERTIES	28
<i>Sample frame</i>	28
<i>Experiment frame</i>	28
DATA PROCESSING	30
<i>Data units</i>	30
<i>Raw data</i>	30
<i>Corrected and normalized data</i>	31
Temperature correction and normalization	31
Profile time correction	32
INJECTION AND BOUNDARY CONDITIONS	33
HYDRODYNAMIC DISPERSION MODEL	33
<i>Analytical solutions</i>	33
<i>Interpretation with the hydrodynamic dispersion model</i>	34
KOVAL'S MODEL	34
<i>Analytical solutions</i>	35
<i>Interpretation with the Koval's model</i>	35
THE COATS AND SMITH MODEL	35
<i>The model</i>	36
<i>Interpretation with the Coats and Smith model</i>	36
STRATIFIED MODEL	37
<i>Model</i>	37
<i>Interpretation with the stratified model</i>	37
GRAPHS	39
RECOMMENDED DATA PROCESSING	40
TUTORIALS	40
TWO-PHASE FLOW MODULE	41
<i>Two-phase flow: direct simulations</i>	42
<i>Fluids and sample properties</i>	42
<i>Capillary pressure: P_c</i>	43
<i>Pc type</i>	43
<i>"Data points" frame</i>	43
<i>Analytical function</i>	44
<i>Amott Index</i>	44
<i>Pc LET function (version 2018)</i>	45
<i>Relative permeabilities: K_r</i>	45
End Points	45
Corey model	46
Modified Corey	46
Kr LET Function	46
<i>Load/fit data Window</i>	47
Raw data	47
Data depending on Saturation	49
Data corrections	49
Correction volumes and flow rates	49
<i>Type of experiment</i>	50
<i>Block times</i>	51
<i>Bond, Capillary, and Rapoport Numbers</i>	51
<i>Inlet Face Capillary Pressure, P_C</i>	52

<i>View (curve display)</i>	52
<i>Local Pressures (version 2014)</i>	52
Loading the abscissas.....	52
Graphs.....	53
<i>Interfacial tension and viscosities values per block time (version 2016)</i>	54
Values	54
Enabling the IFT and viscosities change between block times in simulation.....	54
IFT change implementation	55
Viscosity change implementation	55
NUMERICAL SIMULATION	56
OPTIMIZATION	56
<i>Objective function</i>	57
<i>Optimization per points</i>	58
Moving points	58
Optimization	59
UNSTEADY STATE.....	60
<i>Analytical methods (JBN)</i>	60
Window JBN-Jones and Roszelle	60
<i>Tutorial Kr_USS_MW</i>	61
Starting a project.....	61
Kr calculation.....	62
Simulation with initial guess.....	63
Kr optimization	63
STEADY-STATE	64
<i>Analytical</i>	64
<i>Tutorial</i>	65
Starting a project.....	65
Simulation with first guess.....	66
Kr optimization	66
CENTRIFUGE	67
<i>Local Pc from average</i>	67
Optimization tool	68
<i>Tutorial Pc calculation: Comparison CYDAR – Forbes SCA</i>	68
<i>Relative Permeability – Hagoort</i>	73
The “centri Kr” window, Hagoort part.....	73
Corey fit.....	73
Tutorial Hagoort.....	74
<i>Multistep Kr</i>	75
Speed correction.....	75
POROUS PLATE	77
HETEROGENEITIES.....	77
<i>Heterogeneity data type</i>	77
Homogeneous	78
Composite	78
Length, Permeability, and Porosity	78
Pc curves.....	79
Profiles	79
<i>Capillary pressure</i>	79
<i>Simulation</i>	80
Composite	80
Profiles	80
HYSTERESIS.....	81
Nomenclature and definitions used	81
<i>Pc scanning imbibition curve</i>	82
<i>Pc scanning drainage curve</i>	82
<i>Loop</i>	83
<i>Relative permeabilities</i>	83
<i>Non-uniform initial saturation profile</i>	83
INERTIAL CORRECTION (VERSION 2014)	83
ELECTRICAL METHOD (VERSION 2016)	84
<i>Purpose</i>	84
<i>Principles</i>	84

<i>Experimental setup possibilities</i>	84
Rload at the inlet	85
Rload at the outlet	85
Resistances calculation	85
Contact resistances	85
Saturation calculation.....	86
<i>CYDAR module</i>	86
Electrical method: setup window	86
The experimental data are resistances:.....	87
The experimental data are voltages:.....	87
The current is directly entered	87
The current is calculated from other data:	87
Saturation calculation	87
Input data	87
Output data.....	88
TPF REPORTING	88
<i>Principle of report file</i>	88
<i>Using template:</i>	89
REFERENCES	91

This notice describes the functionalities of the module Core Analysis of CYDAR. The functionalities of CYDAR common for all the modules are described in a separate document (CYDAR Common Features User Manual). The module TPF with EOR is described in a separate document.

All User Manuals are available on our website www.cydarex.fr.

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

CYDAR-SCAL 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 peer-reviewed scientific journals, including the proceedings of the Society of Core Analysis.

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

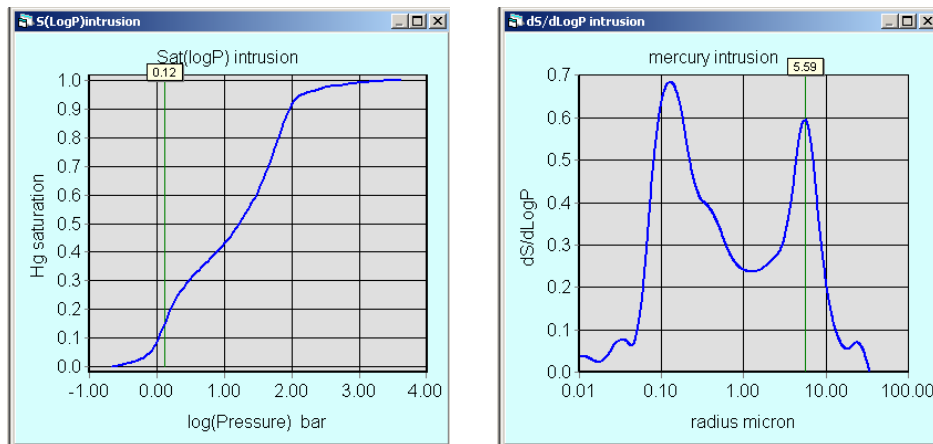
- Pc and pore size distribution, including mercury injection and withdrawal (MICP),
- absolute permeability,
- dispersion,
- two-phase flow experiments,
- steady-state and unsteady-state relative permeability,
- centrifuge capillary pressure and relative permeability.

Pc and pore size distribution (MICP)

The Pc and pore size distribution module 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:

- text files generated by the autopore apparatus version 1 can be opened directly,
- powerful data smoothing using spline functions,
- calculation of three types of pore size distributions,
- calculation of 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.

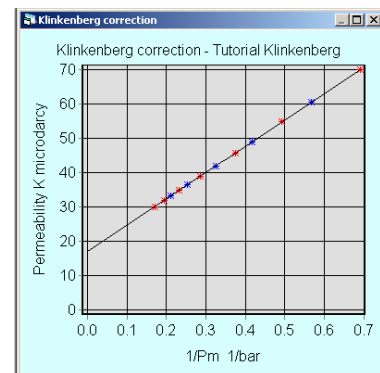


Absolute Permeability

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:

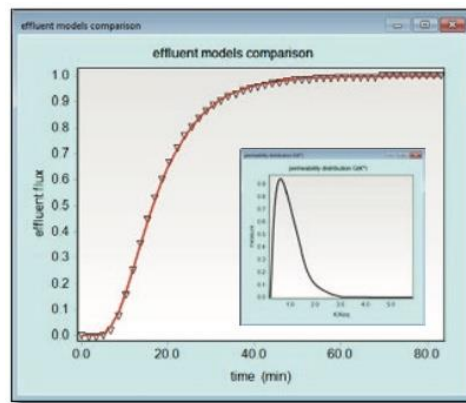
- gas and liquid permeability, steady-state and pulse decay,
- determination of the inertial coefficient (Forchheimer correction),
- determination of the Klinkenberg correction,
- correction for pressure drop in tubings,
- automatic reporting: all results are exported in an Excel file.



Dispersion (CYDAR 2017)

The dispersion / tracer module provides a quantification of the homogeneity of the sample, by analyzing dispersion of an injected solution at concentration C_{in} with a solution at concentration C_0 present in sample.

- 4 models have been implemented: Classical, Koval, Coats and Smith, and Stratified.
- Temperature corrections and normalization are available...



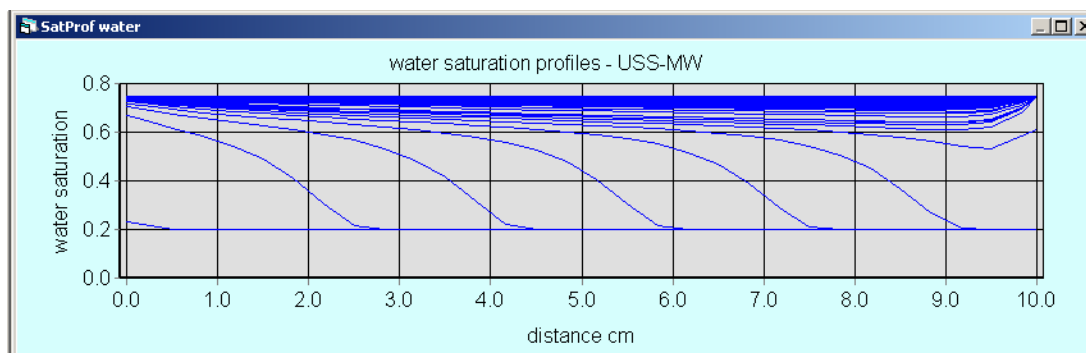
Example of dispersion experiments interpreted using the standard dispersion model (in red) and the stratified model (insert) described in paper SCA2015-014.

Two-Phase Flow Experiment

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

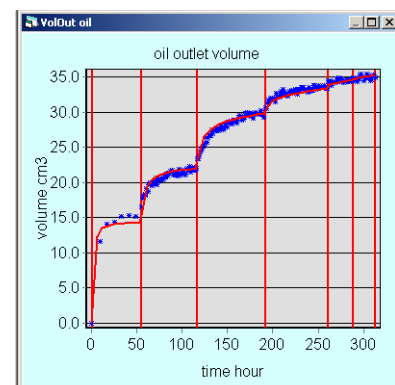
- relative permeability in steady-state and unsteady-state,
- semi-dynamic method,
- spontaneous displacements (immersion of a sample into a fluid in drainage or imbibition),
- gravity flow,
- centrifuge displacements in drainage or imbibition,
- porous plate, with pressure drop inside the porous plates.

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 considered.



In CYDAR, most laboratory experiments are preset, 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, K_r and P_c 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.



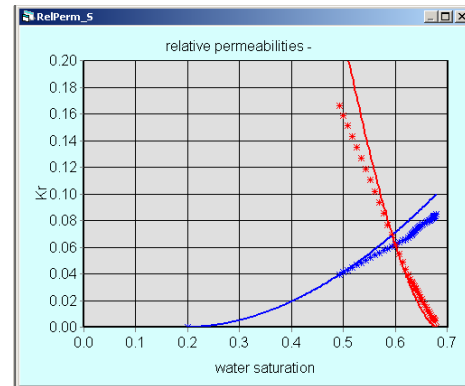
Relative Permeability

Determination of relative permeabilities K_r is one of the main objectives of special core analysis. It is now well recognized that numerical interpretations accounting for capillary effects are an absolute necessity.

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

Main features include:

- K_r 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 K_r 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

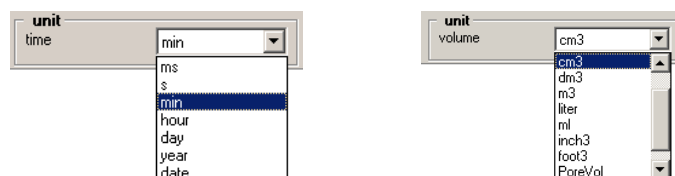
CYDAR is developed in Visual Basic, an object-oriented language, which takes advantage of the familiar Windows environment. Some numerical calculations such as optimization loops use FORTRAN and its powerful IMSL library. However, all FORTRAN DLLs are controlled by CYDAR, and the user interacts only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

CYDAR does not require a high-performance computer, and it runs perfectly on a laptop computer. CYDAR can be installed on Windows operating systems 7, 8, and 10. An Internet connection may be required to activate the license.

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 text 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

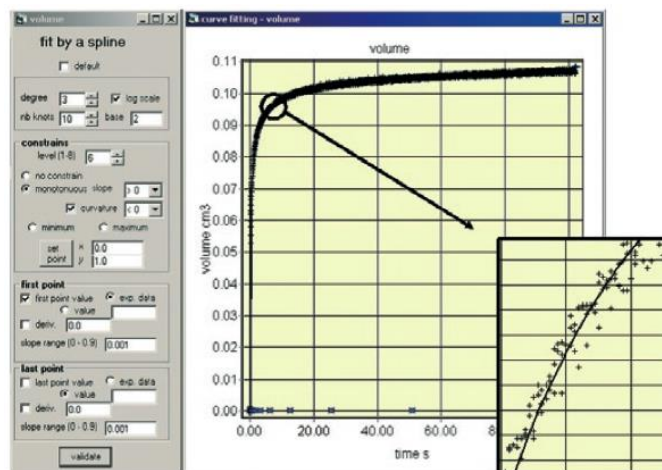
Simulations are performed on a 1-D grid. The size of the grid and parameters for the numerical simulation can be adjusted for speed or accuracy. During numerical simulation and optimization, all variables (flow rates, effluent volumes, pressures, saturations...) are displayed dynamically; and the user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

Exporting results

Graph

From the “export” tab of the graph edition panel (), 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. Data can then be copied and pasted into another software such as Microsoft Excel.

Reporting

Project can be exported as an CSV (comma-separated values) 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
- Pore size distribution 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.

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

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

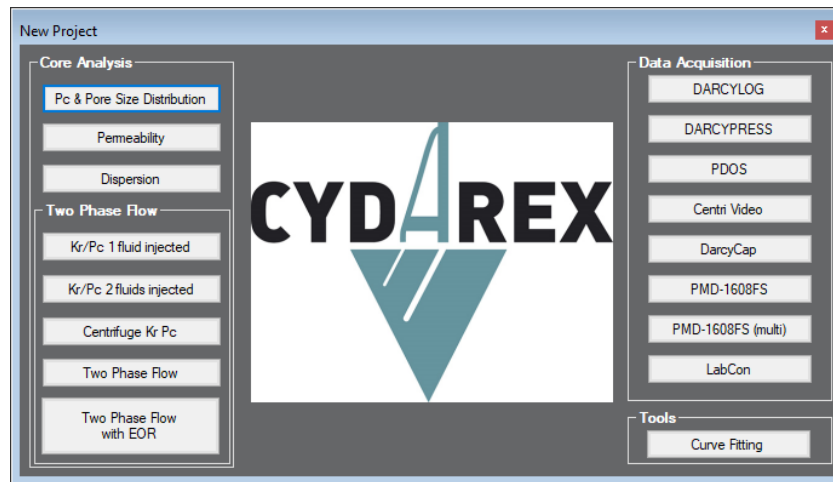


Figure 1: CYDAR new project window.

Pc & Pore Size Distribution: interpretation of experiments with calculation of the Pc curve and pore size distributions ds/dP , psd and $ds/d\log P$ 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.

Dispersion: interpretation of dispersion or tracer experiments, giving indication on the homogeneity of the sample, by analyzing dispersion of an injected solution. Dispersion module may not be activated on your version.

Two Phase flow Module:

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.

Two Phase flow Module with EOR: Adding chemical EOR capabilities on all Two-Phase flow experiments. This module may not be activated on your version.

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

Curve fitting: access to CYDAR curve fitting tools.

Pc & Pore Size Distribution Module

The Pc and Pore Size Distribution module 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 text file generated by the Autopore apparatus version 1,
- 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, Swanson¹¹, and Thomeer¹³ methods,
- automated reporting.

Main window

The “Pc & Pore Size Distribution” button opens the main window (Figure 2). This window allows loading data, fitting curves, calculating PSD (Pore Size Distribution), and more.

Figure 2: Pc & Pore Size distribution 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 a data file from an apparatus (such as Micromeritics Autopore v1), or by entering the volume and pressure data directly. Then click on “load data”.

“**Type of curve**”: Specifies which parts of the data correspond to the first intrusion, to the extrusion, and to the second intrusion (detailed below).

Parameters frame:

Allows to enter the injected fluid (usually mercury) interfacial tension and contact angle under the experimental conditions; these values 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 $ds/d\log P$.

Porosity is calculated from the total volume of fluid 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 Pc curve (Purcell, Swanson¹¹, ...).

Report frame:

- Common to all options: general information, injected fluid and sample properties, experimental data;
- “intrusion” option: $\text{Log}(S_{\text{Hg}})$, $\text{Log}(P)$, r , $dS/d\text{Log}(P)$, S_{ref} and Leverett function;
- “Pc reservoir” option: reservoir and fluids parameters, Pc reservoir, transition zone, water cut.
- “permeability” option: Swanson¹¹ 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.

Loading experimental data

Load data can be imported in 3 different ways:

1. Automatically loading the Autopore version 1 file with a .rpt extension. Once selected in the dropdown menu, press load data and a Open File window will open. Select the appropriate file and press “Open”.
2. Loading the cumulative volume as a function of the injection pressure. The porosity is then calculated from the maximum injected volume and the sample “total volume”.
3. Loading the cumulative volume per gram as a function of the injection pressure. The porosity is then calculated from the “sample mass” and the “bulk density”. The sample “total volume” is then inactive and the total volume value is calculated from the sample mass and bulk density. Note that the sample mass needs to be entered before entering the volume per gram data.

Types of curves

The “type of curve” window (Figure 3) 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 (for example, for conformance corrections or surface effects in mercury injection). The volumes for saturation calculation can be either taken from the intrusion first and last points (checkbox), or entered manually.

Figure 3: “type of curve” window.

Pore size distribution

pore size distribution $f(r) = \frac{p^2}{2\gamma \cos \theta} \frac{dS}{dP}$	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 1: 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 4): S(LogP), dS/dP, psd and ds/dLogP.

In the view window (right side of Figure 4) the frames called “intrusion”, “extrusion” and “2nd intrusion” are enabled if the corresponding types of curve have been selected in the “type of curve” window (Figure 3).

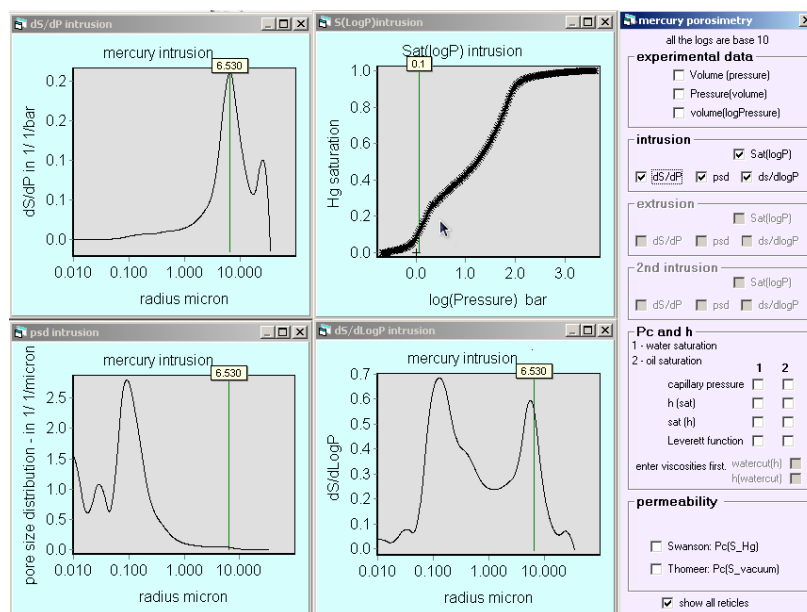


Figure 4: Graph examples in mercury injection.

Permeability calculation

Permeability can be estimated from the pore size distribution and porosity. The “tube model”, Katz and Thompson, Swanson¹¹ and Thomeer¹³ relationships are implemented. The porosity value is necessary for all models.

The screenshot shows the 'Permeability' software window. At the top, it states 'permeability is calculated for intrusion only'. Below this, there are input fields for 'porosity' (0.113), 'frac.' (a dropdown menu), and 'mdarcy' (a dropdown menu). The 'Models Based On Formation Factor' section includes a checked box for 'default: phi ^ - 2', a 'tube model' input (34.005), and a 'Katz Thompson' input (42.609). The 'Pc Interval' section has a range from 0.500 to 15.000 with a unit of 'bar'. The 'Swanson' section contains a table of results: Saturation (0.529), S_bulk (5.976), Pc (0.772), and radius (9.625). It also has a 'result' column and a 'graph' checkbox. The 'Thomeer' section includes a table of results: frac Vol at P_inf (0.117), Pd (entry Pc) (0.279), and G (0.292). It also has a 'result' column, a 'permeability' input (165.247), and a 'Fit' button.

Figure 5: 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 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 6).

Thomeer:

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

The “Fit” button opens the “modified hyperbolic” window (

This screenshot is identical to the one in Figure 5, showing the 'Permeability' software window. The 'Fit' button in the Thomeer section is highlighted, indicating it is the next step in the procedure.

- Figure 5) 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 7).

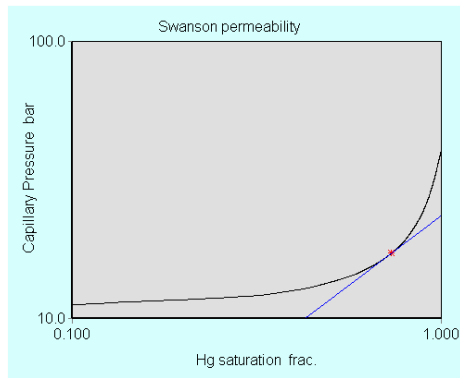


Figure 6: Swanson method: automatic calculation of the tangent.

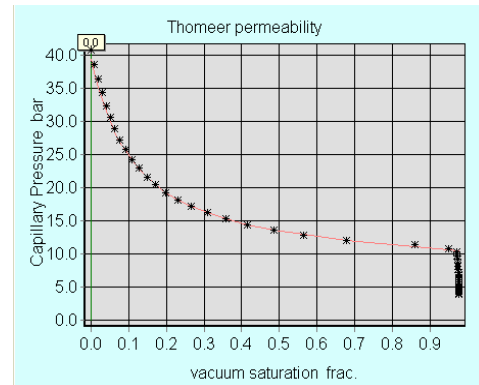


Figure 7: Thomeer method: fit of the $P_c(\text{sat})$ curve by an hyperbolic function in log/log scale.

Capillary pressure

The “Capillary pressure and Leverett function” window (

Figure 8) is used to enter parameters for the J Leverett function and P_c 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 capillary pressure (M.C. Leverett. "Capillary behavior in porous solids". *Transactions of the AIME* (142): 159–172, 1941):

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

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

Figure 8: 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 P_c 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 ($>90^\circ$).

Transition zones

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

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

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

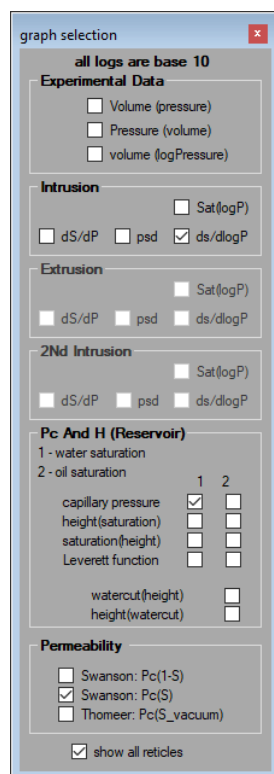
Water cut

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:

$$f_w = \frac{Q_w}{Q_w + Q_o} = \frac{K r_w / \mu_w}{K r_w / \mu_w + K r_o / \mu_o}$$

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

Displaying graphs



Graphs can be displayed from the “graph selection” window, from the View menu. When graphs cannot be calculated, mostly due to the lack of data, the corresponding checkbox is not active.

Experimental Data frame displays the experimental data entered with the “load data” function.

The different pore size distributions can be displayed from the Intrusion, and when data are available, from the Extrusion and 2d Intrusion.

The “Pc and h (reservoir)” frame displays the Pc, the height saturation, Leverett function, and watercut data.

The Permeability frame shows the Swanson and Thomeer data.

Tutorial files

Tutorial_Hg_SampleA

This example uses a text file generated by an AUTOPORE apparatus version 1 (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_Brauwillier

This example uses data from the Microsoft Excel spreadsheet “Tutorial_Hg_Brauwillier.xls”.

To calculate the porosity, the user must enter the total volume of 7.733 cc.

The corresponding CYDAR project is “Tutorial_Hg_Brauwillier.cyd”.

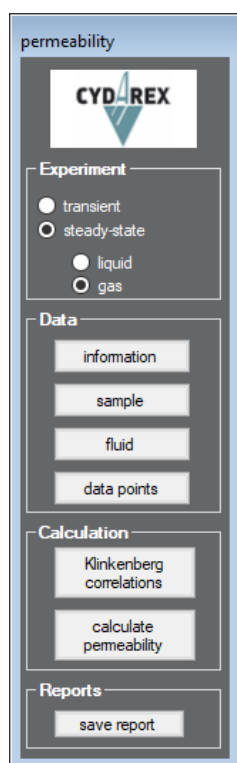
Permeability Module

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



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”)

The **“save report”** button generates an automatic report of the analysis in tab-separated values, which can be opened in Microsoft Excel.

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 10). Because the porosity can be optimized in the transient module, its input is on the calculation window (Figure 9). Figure 11 shows the fluid properties windows.

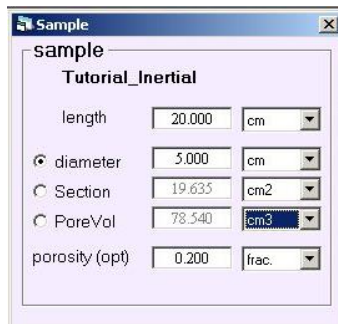


Figure 10: Sample window.

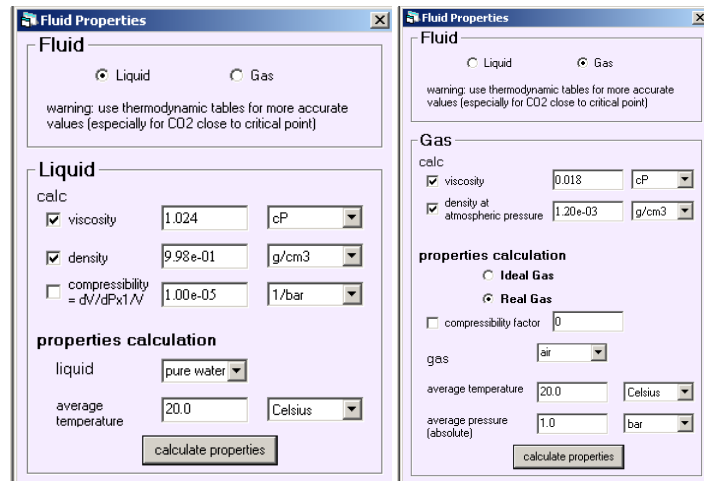


Figure 11: "Fluid properties" windows.

Klinkenberg correlations

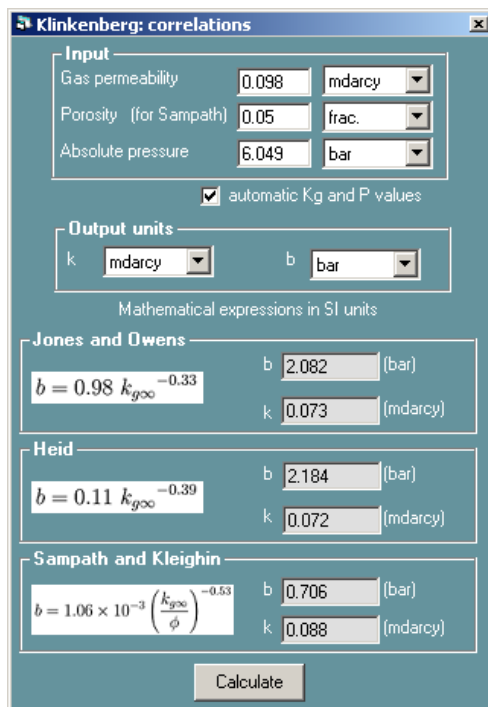


Figure 12: Klinkenberg correlations window.

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 9) 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 9 opens a spreadsheet (Figure 13 and Figure 14). 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 “**P_{inlet}-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 P_q 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.

date	hour	DP	Q (volume)	symbol 0-1	comments
1		0.690	0.006	1	
2		0.840	0.008	1	
3		1.220	0.013	1	
4		1.400	0.014	1	
5		1.490	0.018	1	
6		2.300	0.023	1	
7		2.300	0.024	1	
8		2.630	0.026	1	
9		2.690	0.030	1	
10		3.370	0.034	1	
11		3.670	0.036	1	
12		3.880	0.038	1	
13		4.490	0.043	1	

Figure 13: Data points spreadsheet for a liquid.

date	hour	P _{in}	Q (volume)	symbol 0-1	comments
1		0.862	36.000	1	
2		2.023	120.000	1	
3		3.303	222.000	1	
4		4.980	394.000	1	
5		6.584	558.000	1	
6		8.222	756.000	1	
7		9.722	860.000	1	
8		7.403	854.000	0	
9		5.878	474.000	0	
10		4.125	294.000	0	
11		2.765	180.000	0	
12		1.496	84.000	0	
13					

Figure 14: 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 15) 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:

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

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

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

Figure 15: Permeability calculation.

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 $(P_{in}^2 - P_{out}^2) / (2P_q L)$. 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:

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

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:

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

The “**calculate permeability**” button opens Figure 15.

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.

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

$$\frac{(P_{in} - P_{out})}{L} = \frac{\mu U}{K} (1 + BRe)$$

Typically, the inertial effect becomes significant when $Re \geq 1$. 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:

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

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 $K(1/\langle P \rangle)$ 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

The screenshot shows the 'Data Points unsteady state' window. It includes a schematic of a two-volume system with pressures P_1 , P_2 and volumes V_1 , V_2 , and dead volumes V_{d1} , V_{d2} . The window is divided into several sections for data entry:

- Inlet dead volume:** dead volume V_{d1} and V_{d2} (cm³).
- Reservoir volumes:** V_1 (at Patm) and V_2 (at Patm) (cm³).
- Compressibilities ($dV/dP \times 1/V_{atm}$):** V_1 compressibility, V_2 compressibility, and Cell compressibility (1/bar).
- P atmospheric (absolute):** (bar).
- Experimental set up:** ☒ 2 volumes, ☐ volume-pressure.
- Initial sample pressure (relative):** ☒ P1, ☐ P2, ☐ Steady-State.
- P1 (relative):** ☒ initial pressure (0.000 bar), ☐ initial pressure from data file.
- P2 (relative):** ☒ initial pressure (0.000 bar), ☐ initial pressure from data file.
- Experimental data:**
 - Corrected pressures:** ☒ P1 (relative), ☐ P2 (relative), ☐ DP.
 - Raw data:** ☐ P1 (relative), ☐ P2 (relative), ☐ DP.
- Pressure offsets:** (button).
- Fit/Edit:** load data points, view/modity, remove, Edit, linear (dropdown), fit.

Figure 16: 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. 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

The screenshot shows a software window titled "permeability pulse decay" with a sub-header "tutorial PulseDecay". It contains several sections for parameter input:

- fit**: A table with columns for fit, initial, final, unit, and bounds. It includes checkboxes for $K_{P \rightarrow \infty}$, ϕ , b , and β , each with corresponding numerical input fields and unit dropdowns.
- Corrections**: Checkboxes for "Klinkenberg" and "Inertial".
- data points to fit**: Radio buttons for "P1", "P1 and P2", "P2", and "Dp".
- Cost Function Calculation**: A "time interval" input field with a unit dropdown (set to "s").
- Simulation time (%)**: An input field.
- Root Mean Square Error**: An input field showing "0.00e+0".
- graph convergence**: A checkbox.
- Show curves during run (slower)**: A checkbox.
- Buttons**: "run no optim", "run optim", "accept", "stop", and "save simulation".

Figure 17: Calculation panel for transient permeability.

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

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

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

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

The numerical scheme is implicit and the system is solved by a Newton-Raphson algorithm. The cost function is calculated with P₁, P₂, both, or DP=P₁-P₂ (frame “data points to fit” in Figure 17).

Buttons:

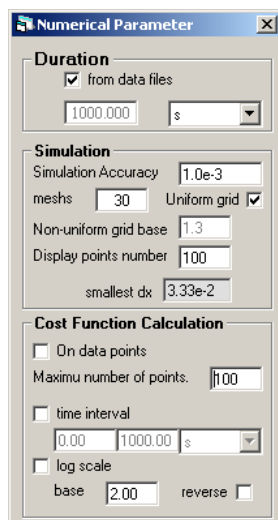
- “**run no optim**”: runs one simulation without any optimization.
- “**run optim**”: runs optimization on selected parameters: permeability, porosity, Klinkenberg coefficient and/or inertial coefficient.
- “**accept**”: accepts optimized values. This action erases the first guess values.
- “**stop**”: 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 18):



“**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 18: 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_Inertial.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”.

Dispersion Module

The Dispersion Module is launch from the New Project window, by selecting “Dispersion.” If you are running the Dispersion module in Demo mode, you might be prompt to enter an activation passcode. Contact Cydarex to obtain this code.

Dispersion main window

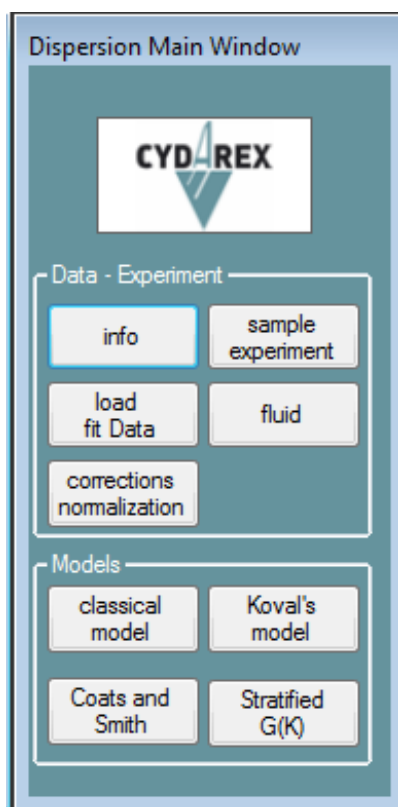


Figure 19: Main window of dispersion module.

Data - Experiment frame:

“Info”: common to all modules, allows entering information regarding the sample and experience. None of this information is used for calculation.

“sample/experiment”: dimensions and porosity of a cylindrical sample, injection, effluent, and profiles data type (concentration or flux), Reynolds and Peclet numbers.

“load/fit data”: input data spreadsheet, data corrections, possibility to edit and fit data.

“Fluid”: fluid properties window, used only for the Reynolds calculation.

“corrections/normalization”: temperature correction and normalization of the data, time correction for the profiles.

Models frame:

“Classical model”: fit with the “classical” dispersion model.

“Koval’s model”: fit with the Koval’s model.

“Coats and Smith”: Simulation and optimization using the Coats and Smith model.

“Stratified G(K)”: interpretation of the experiment with a stratified medium giving a distribution of permeability.

Sample and experiment properties

Sample frame

The sample is assumed to be cylindrical. The length and either the diameter, the section, or the pore volume if the porosity is known, need to be entered (Figure 20). The permeability is optional and is only used to calculate the Reynolds and Péclet numbers displayed in the experiment frame.

Experiment frame

First the user chooses if the injected concentration is greater than the initial concentration in place $C_{in} > C_0$ or the opposite situation $C_0 > C_{in}$. The flow rate is needed for calculation. The diffusion coefficient is optional and used only for calculation of the Péclet number.

The Péclet number represents the ratio of advective transport over diffusive transport, and is defined as follow:

$$Pe = \frac{V\sqrt{K}}{D_m},$$

where V is the front velocity $V = Q/(A\Phi)$, with A the sample section, Q the flow rate, and ϕ the porosity. K is the permeability and D_m is the coefficient of molecular diffusion, with a default value of $2 \cdot 10^{-5} \text{ cm}^2/\text{s}$.

The Reynolds number is the ratio of inertial forces to viscous forces, and is defined as follow:

$$Re = \frac{\rho Q \sqrt{K}}{A \Phi \mu},$$

where ρ and μ are the density and viscosity of the fluid.

Finally, the user selects what are the effluent and profiles data type: flux or concentration. Practically the flux is measured at the outlet, and the profiles are concentrations.

Figure 20: Sample window

Data processing

As in other modules (see CYDAR User Manual Common Features), there are two main groups of data:

- Raw data are the direct experimental data that have been uploaded.
- Corrected and normalized data are the data used for interpretation. These data can have been corrected or normalized, and have been fitted.

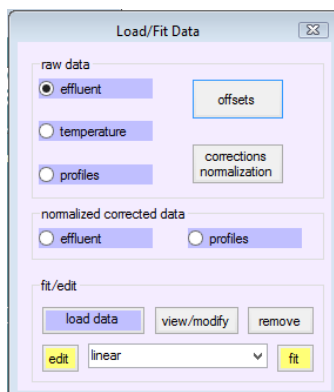


Figure 21: Load/Fit Data window, allows performing data offset correction, editing, etc...

The “Load/Fit window” (Figure 21) is used to:

- Load data.
- Open the “Data offset corrections” window (Figure 22).
- Open the “temperature correction and normalization” window (Figure 23).
- Edit and perform fit. A fit needs to be performed for each data set. A linear fit is the default fit, and the “fit” button needs to be pressed.

Data units

All interpretations are performed on data that are assumed to be normalized. Therefore, units of the measurements have no impact on the interpretation. For instance, the effluent can be a tension or a density measure, the profiles can be X ray scan or resistivity measurement; in all cases the normalization will lead to dimensionless physical quantities:

$$Y^* = \frac{Y_{exp} - Y_{min}}{Y_{max} - Y_{min}}$$

Y^* is the normalized dimensionless physical quantity. Y_{exp} is the measurements of this physical quantity (tension, density, X ray, etc.). Y_{max} and Y_{min} are the extrema with the same unit than Y_{exp} used for normalization. See below to know how to perform normalization and correction.

The other physical values, like temperature, time, and distance, have the proper unit type.

Raw data

There are three different data sets:

- Effluent: measurements versus time.
- Temperature: temperature versus time.
- Profiles: a set of measurements at different locations along the sample performed at different time.

The effluent and profiles raw data cannot be edited, but they are used to calculate corrected and normalized data. These data can be edited. In CYDAR, profile data cannot be edited.

In CYDAR, the corrected data are the data used for calculation and interpretation; see the recommended way to process data below.

The temperature data is used for measurements correction (see below). The calculation uses the analytical fit of the data. Therefore, any edition and fit must be done before calculation of the correction.

Offset correction, due to dead volume for example, can be performed from the “Data offset corrections” window (Figure 22). This window is open with the button “offsets” on window “Load/Fit window” (Figure 21).

Figure 22: Data offset correction window.

Corrected and normalized data

Data are calculated from the effluent and profiles raw data. Figure 23 shows the dedicated window with the different entries.

Temperature correction and normalization

The temperature correction and normalization are performed on both effluent and profiles data. The different parameters units are assumed to be similar to the data's units. All parameters must be set before calculation by pressing the “calculation” button.

Figure 23: Temperature correction, normalization, and time correction for profiles window.

The effect of the temperature correction can be checked on the graph with the raw data. These graphs can be opened in the “Graphs” window: “effluent” and “profiles” in frame “experimental data” (Figure 33).

The Figure 24 shows an example of the effluent graph displaying the raw data (in black), the correction by temperature before normalization (in green) and the minimum and maximum value used for normalization (horizontal lines). Any change in the parameters will update graphs.

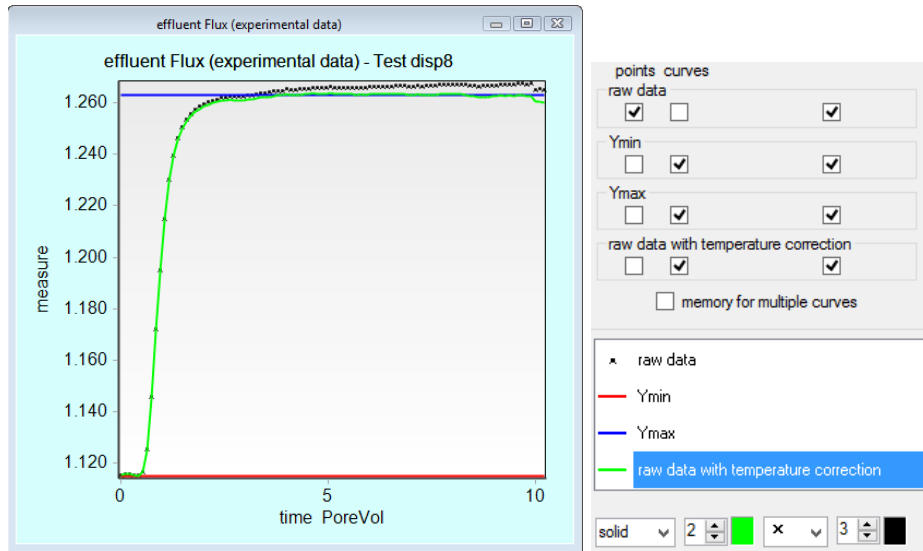


Figure 24: a graph with an example of the temperature correction effect and the normalization extrema displayed. The raw data is in black, the green curve is the corrected data; the blue and red horizontal lines are Y_{min} and Y_{max} .

When the correction and normalization are done, the calculated data can be displayed on the graphs of the “Models (Fit)” section of the “Graph” window (Figure 33). All the graphs in this section display these calculated data as “corrected and normalized data”. After correction, a fit needs to be run.

The temperature correction is given by:

$$Y_{cor} = Y_{exp} - b(T - T_0) - a(T - T_0)^2$$

Y_{cor} is the corrected measurement before normalization. Y_{exp} is the raw data. T_0 is the reference temperature. And a and b are coefficients with proper units.

The normalization is simply:

$$Y^* = \frac{Y_{cor} - Y_{min}}{Y_{max} - Y_{min}}$$

Profile time correction

A profile is a set of measurements along the sample started at a given time. Depending on the setup, these measurements cannot be simultaneous but sequential. In other words, the profile is not an instantaneous photography of the experiment. It can be useful to account for the duration of the measurements.

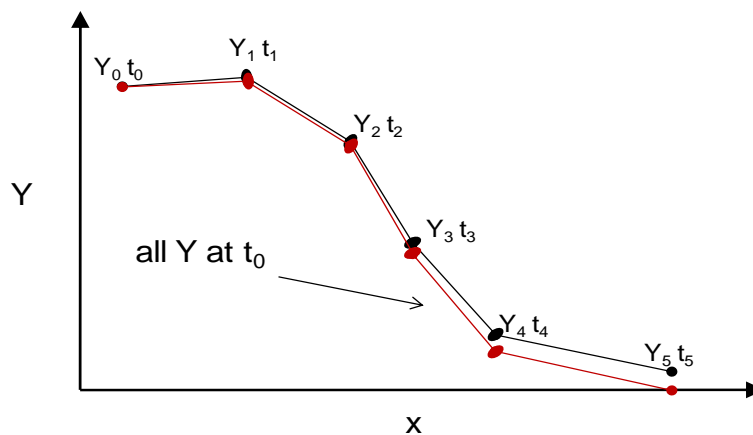


Figure 25: sketch of the measuring duration effect on a profile measurement.

Figure 25 shows the effect on the profiles measurements when the duration of measurements is not negligible. In this hypothetic experiment, one profile measurement was started at t_0 . The points were

taken sequentially at different increasing time: t_1 to t_5 . The red line would be the profile obtained with simultaneous measurements.

The dispersion module offers the possibility to account for such effect. In practice, the corrected and normalized experimental data are not modified in CYDAR. But the models, simulations, and calculations account for the correction. For instance, when a fit is performed with a model the optimization of the parameters is done on corrected profiles. And the fit curves are displayed with the same time than the experimental one.

The user enters the average measuring duration in the “temperature correction and normalization” window (Figure 23). The user chooses also what the times for profiles loaded in the spreadsheet are. “Experimental profiles times are”:

- “times of the first records”
- at half of the measuring duration
- “times of the last records”.

Injection and boundary conditions

Very often experiments with a continuous injection with a constant concentration C_{in} are interpreted using analytical solution obtained assuming constant concentration at the inlet. However, the physical boundary condition is a constant density flux. Experiments on micromodels have shown that concentration at the inlet of the porous medium is not constant. Practically we inject a fluid at a constant concentration, but this is the flux density that is constant, see for instance the 2015 SCA paper from Lenormand et al. (Lenormand, Lenormand, & Baugé, 2015).

Therefore, the interpretation in CYDAR assumes a continuous injection of a constant flux from time $t=0$:

$$F_{in}(t) = QAC_{in}u(t) \text{ with } u = \begin{cases} 0 & t < 0 \\ 1 & t = 0 \\ 1 & t > 0 \end{cases}$$

$F_{in}(t)$ is the flux density which is zero before the start of the injection then constant during the experiment. C_{in} is the concentration of the injected fluid. Q is the volume flow rate that is constant as well. And A is the sample section. C_{in} may either be smaller or greater than the initial concentration C_0 in place. The entries to specify the injection properties are on the window “Sample – Experiment” (Figure 20), see below.

The outlet has no effect on the spreading in the sample. The analytical solutions are obtained assuming a zero concentration flux at the infinite. The outlet boundary condition used in the simulation for the Coats and Smith model is a zero second derivative of concentration according to space: $\partial^2 C / \partial x^2 = 0$.

Hydrodynamic dispersion model

Analytical solutions

The model corresponds to:

$$F = V\phi \left(C - \alpha \frac{\partial C}{\partial x} \right),$$

$$\frac{\partial C}{\partial t} + \frac{1}{\phi} \frac{\partial F}{\partial x} = 0,$$

with F the flux density, C the concentration, V the front velocity ($V = Q/(A\phi)$), α the dispersivity.

With the boundary conditions given above, a constant flux injection and a zero concentration at the infinite, the analytical solutions for the flux and concentration are:

$$f = \frac{f_{in}}{2} \left(\operatorname{erfc}(z_1) + \exp\left(\frac{x}{\alpha}\right) \operatorname{erfc}(z_2) \right),$$

$$C = \frac{C_{in} V}{2} \left(\operatorname{erfc}(z_1) - \left(1 + \frac{1}{\alpha} (x + Vt) \right) \exp\left(\frac{x}{\alpha}\right) \operatorname{erfc}(z_2) + 2 \left(\frac{Vt}{\pi \alpha} \right)^{\frac{1}{2}} \exp(-z_1^2) \right),$$

with

$$z_1 = \frac{x - Vt}{2(\alpha Vt)^{1/2}},$$

and

$$z_2 = \frac{x + Vt}{2(\alpha Vt)^{1/2}}.$$

See for instance (Lenormand, Lenormand, & Bauget, 2015) or equation 13 of the Coats and Smith 1964 paper (Coats & Smith, 1964).

Remark: the concentration equation given in the SCA paper contains a typo: the factor V and 2 are missing.

Interpretation with the hydrodynamic dispersion model

The dedicated window is open by clicking the button “model classic”. Figure 26 shows this window.

In the frame “graphs” the user can open or close some graphs where the result of the fit can be displayed. The fit can be done on effluent and/or profiles data, the choice is done using the corresponding checkboxes. The user can also choose which profiles will be used for the fit calculation (group of checkboxes in the window bottom).

The fit is performed by optimizing the dispersivity. The textbox with a white background is the initial guess, and the one with a light grey background is the value corresponding to the best fit.

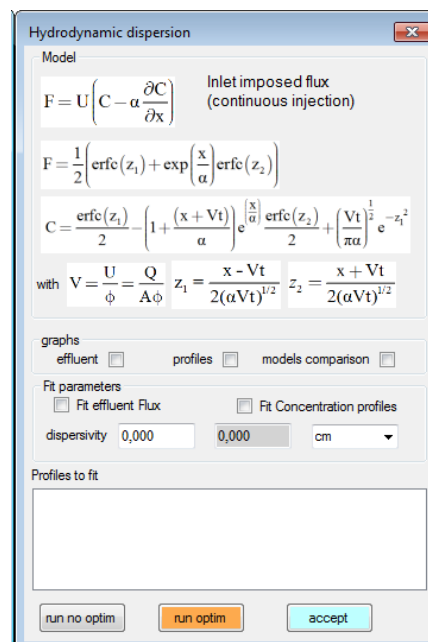


Figure 26: the “Hydrodynamic dispersion” model window.

The “run no optim” button allows to calculate the analytical solutions for the effluent and profiles using the initial guess without any optimization. The “run optim” launches the fit. The “accept” button passes the value of the best fit to the initial guess textbox and set this value as the current dispersivity.

Remark: only the value displayed in the initial guess textbox is saved in the project file.

Koval’s model

The Koval’s model is a one-dimensional homogenized model of viscous fingering in miscible flow. It not only accounts for viscosities ratio effect but also for the medium heterogeneities through a heterogeneous factor H, see details in the original paper of Koval (Koval, 1963).

In the dispersion module of Cydar, only the heterogeneity factor is considered. There is no viscosity ratio.

Analytical solutions

The model is:

$$F = \frac{HC}{C(H-1)+1},$$

$$\frac{\partial C}{\partial t} + \frac{1}{\phi} \frac{\partial F}{\partial x} = 0.$$

The flux solution is:

$$f = \begin{cases} 0 & t \leq \frac{x}{Vt} \\ \left(H - \frac{\sqrt{Hx/(Vt)}}{H-1} \right) & \\ 1 & t \geq \frac{xH}{V} \end{cases}$$

And the concentration solution is:

$$c = \begin{cases} 1 & x \leq \frac{Vt}{H} \\ \left(\frac{\sqrt{HVt/x} - 1}{H-1} \right) & \\ 0 & x \geq HVt \end{cases}$$

Interpretation with the Koval's model

Figure 27 shows the Cydar's window used for interpretation with Koval's model.

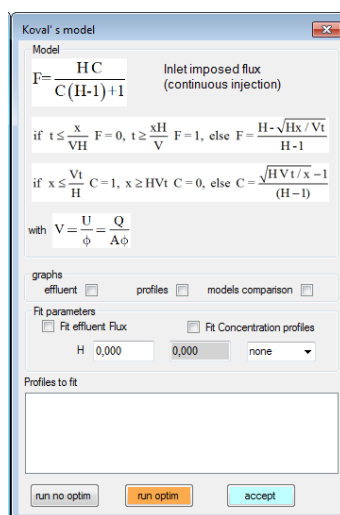


Figure 27: the "Koval's model" window.

Entries and principle are the same that with the hydrodynamic dispersion model. The only difference is that the optimized parameter is the heterogeneity factor.

The Coats and Smith model

In porous media, dead volumes may exist and can lead to long time tail behavior. Coat and Smith have adapted the hydrodynamic model (see above) to this kind of behavior by adding a mass transfer equation between the flowing and the stagnant regions (Coats & Smith, 1964).

The model

The flux-concentration relation is the same than in the hydrodynamic dispersion model:

$$F = V\phi \left(C - \alpha \frac{\partial C}{\partial x} \right).$$

In the mass balance equation, a new term accounting for the non-flowing fraction of fluid is added:

$$(1 - S_f) \frac{\partial C_s}{\partial t} + S_f \frac{\partial C}{\partial t} + \frac{1}{\phi} \frac{\partial F}{\partial x} = 0,$$

$$(1 - S_f) \frac{\partial C_s}{\partial t} = K(C - C_s),$$

with C_s the concentration in the non-flowing fraction of fluid, S_f the fraction of flowing fluid and K the rate of transfer between flowing and stagnant volumes.

Interpretation with the Coats and Smith model

Figure 28 shows the “Coats and Smith model” window. The optimization is done on the effluent data. Three parameters can be adjusted: the dispersivity, the transfer rate and the flowing fraction. The textboxes with the white background are the values used for simulation and the initial guesses of the optimization. The boxes with the light grey background are the final value of the optimization. The root mean square error is indicated. The user chooses which parameter to optimize using the checkboxes on the left.

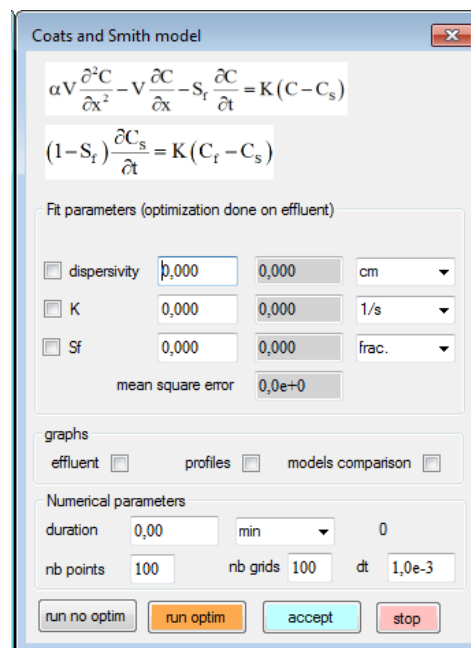


Figure 28: the “Coats and Smith model” window.

The numerical parameters are:

- the simulation duration
- the number of outputs asked (“nb points”), asked because some points may be added to correspond to the times for profiles
- the number of grids (“nb grids”)
- the dimensionless time step “dt”

The buttons:

- “run no optim” starts one simulation using the values entered in the textboxes with white background
- “run optim” launch the optimization on the selected parameters

- “accept” passes the optimized values to the initial guesses
- “stop” stops running simulation and optimization.

As for the other model, only the value displayed in the initial guess textbox is saved in the project file.

Remark: The simulation data are not saved, only the parameters displayed on the model window are saved. Therefore, to display the simulation outputs on graph, the simulation must be rerun after the opening of a project.

Stratified model

Model

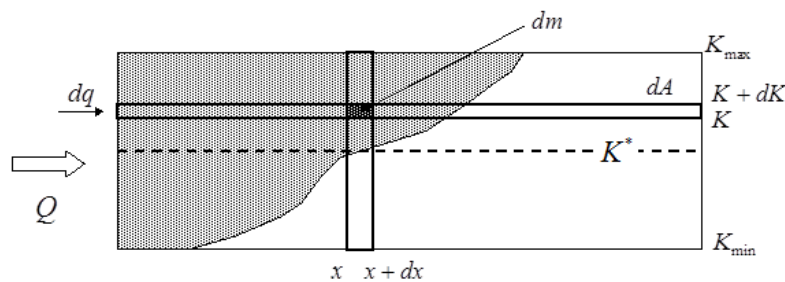


Figure 29: Perfectly layered medium: principle of the calculation of tracer transport.

Stratified media have been extensively used as model of heterogeneous media either for immiscible or miscible flow. The heterogeneous porous medium is represented by a perfectly layered medium with a permeability distribution function $G(K)$. Microscopic dispersion is assumed to be negligible, and the tracer follows the streamlines. To simplify the calculation, the layers are re-arranged with increasing permeability (Figure 29). At a given time, the front of tracer is calculated using the flow rate in each layer and the concentration, by averaging over the cross-section:

$$C(x, t) = C_0 \int_{K^*}^{K_{max}} G(K) dK,$$

where $K^* = \frac{K_{eq}}{V} \frac{x}{t}$ is the lower limit of the invaded layers and K_{eq} is the effective permeability. The flux at the same position is calculated by summing the flow rates in all invaded layers:

$$F(x, t) = \int_{K^*}^{K_{max}} C_0 \, dq = C_0 Q \frac{1}{K_{eq}} \int_{K^*}^{K_{max}} K \, G(K) dK.$$

The curve can be determined numerically by elimination of the parameter K^* .

The dispersion module of CYDAR allows the calculation of $G(K/K_{eq})$ from the profiles and the effluent data, and the calculation of the profiles from these permeability distributions.

Interpretation with the stratified model

Figure 30 shows the “stratified model” window. The beta coefficient is a parameter allowing to account for non-convective spreading.

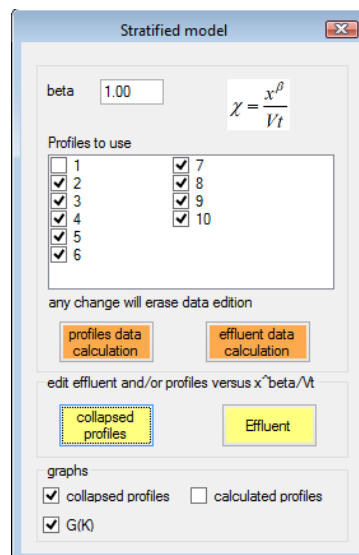


Figure 30: "Stratified model" window.

The orange buttons "profiles data calculation" and "effluent data calculation" launch the calculation of the collapsed profiles and the effluent versus $x^\beta/(Vt)$. The collapsed profiles are all the selected profiles grouped in one data that can be fit using splines for instance. Figure 31 shows an example of a graph displaying calculated collapsed profiles, its fit and the fit of the effluent data.

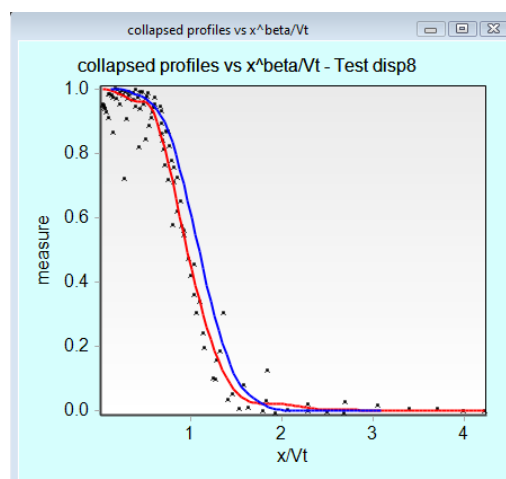


Figure 31: "collapsed profiles" graph displaying an example of calculated collapsed profiles (black dots), its fit (red line) and the fit of the effluent data versus $x^\beta/(Vt)$ (blue line).

The yellow buttons on the "stratified model" window open the "edit points panel" for the collapsed profiles and effluent data from which editing and fitting can be performed.

The two permeability distributions are calculated from the fits of the collapsed profiles and the effluent versus $x^\beta/(Vt)$.

Any new calculation erases previous data edition and existing fit.

The checkboxes in the "graphs" frame open the corresponding graphs:

- "collapsed profiles" opens the graph where collapsed profiles data, its fit and the fit of effluent data versus $x^\beta/(Vt)$ can be displayed
- "G(K)" is the graph allowing to display the two permeability distributions versus K/K_{eq} calculated from the collapsed profiles fit and the effluent fit (Figure 32)
- on "calculated profiles" the user can display the profiles calculated from these G(K)

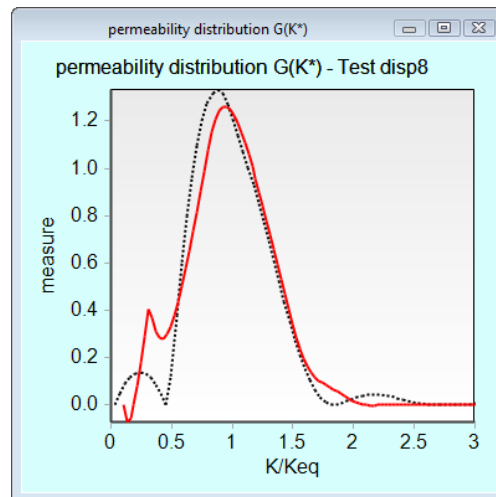


Figure 32: example of permeability distributions calculated from profiles (dot line) and effluent (red line).

Graphs

As in other CYDAR modules, the menu “view” opens a window from which the different graphs can be open or closed (Figure 33).

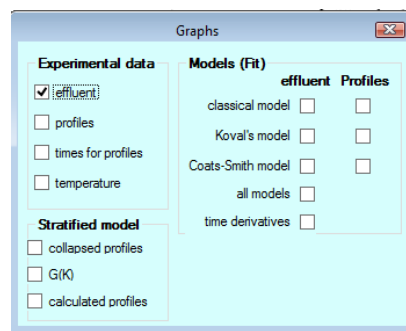


Figure 33: “Graphs” window allowing to display the different graphs.

Frame “Experimental data”:

- “effluent”: graph with the effluent data loaded and before any correction, the extrema used for normalization and the result of temperature correction before normalization;
- “profiles”: idem with the profiles data;
- “times for profiles”: the times loaded with the profiles;
- “temperature”: the temperature data used for the temperature correction.

Frame “Models (Fit)”:

- For each model: graphs with the normalized and corrected data for “effluent” and “profiles” and the output of the corresponding model;
- “all models”: graph with the comparison of the three effluents calculated;
- “time derivative”: graph with effluent time derivatives of the normalized data and the calculated data.

Frame “Stratified model”:

- “collapsed profiles”: graph with collapsed profiles data, its fit and the fit of effluent data versus $x^\beta / (Vt)$;
- “G(K)”: graph with the two permeability distributions versus K/K_{eq} calculated from the collapsed profiles fit and the effluent fit;

- “calculated profiles”: graph with the profiles calculated from these $G(K)$.

Recommended data processing

1. Entering the general information
2. Entering the sample properties and experimental characteristic
3. Entering the fluid properties (used only for the Reynolds calculation)
4. Loading the raw data:
 - a. no editing for the effluent and profiles
 - b. editing the temperature if necessary because this is its analytical fit which will be used for correction
5. Performing the offsets correction, for instance that is the way to take into account the dead volume effect on the effluent measurement
6. Performing the normalization, the temperature correction and the time correction for the profiles if desired, giving the data used for interpretation
7. Performing interpretation with the different models

Tutorials

Tutorials files can be downloaded from CYDAREX website. For the module Dispersion, the following files are available:

- Dispersion_Tutorial_SCA8.xlsx
- SCA_8_effluent.cyx
- SCA_8_profiles.cyx

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 (K_r) and capillary pressure (P_c) are assumed to be known.

Parameter estimation: To use experimental data to determine unknown parameters, mainly K_r and P_c . 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 K_r and P_c :

- **Unsteady-state** experiments for K_r determination. One fluid is injected and the transient productions and pressures are used to determine K_r . 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 K_r 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 K_r , 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

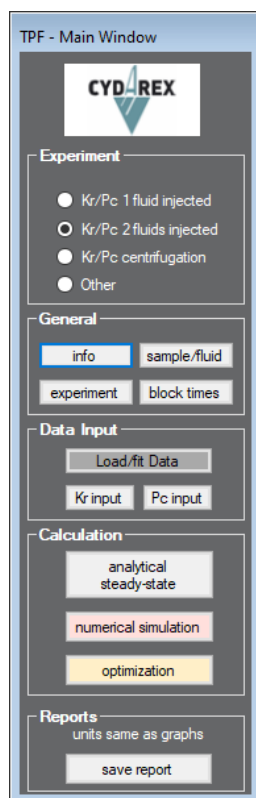


Figure 34: two-Phase Flow (TPF) main window

Experiment 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 42).

“**experiment**” opens the experiment type settings window (page 50) allowing various type of condition: centrifuge, one or two fluids injection, spontaneous, etc.

“**block times**” (page 51) 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 43).

“**Kr input**” opens the relative permeabilities panel with fit, Corey model, Kr LET, JBN calculation, etc (page 45).

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).

“**save report**” opens a dialog window to export project as a tab separated file, which can be open in Microsoft Excel.

Fluids and sample properties

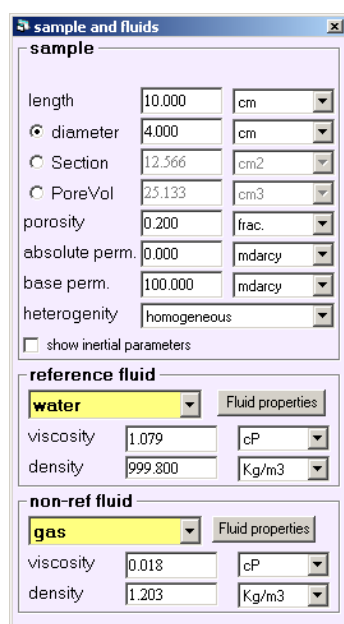


Figure 35: Two-phase flow's "sample and fluids" properties window.

Sample properties:

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

There are three kinds of heterogeneity:

- **Composite with one Pc curve** means a sample made of several homogeneous blocks with only one Pc curve.
- **Composite with multi Pc curve** means a sample made of several homogeneous blocks and each block has its own Pc curve.
- **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 77) 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 83 for detail).

The “**Fluid properties**” buttons give access to the fluid properties window described previously (Figure 11).

The viscosity and density are entered for the reference and non-reference fluids. Per definition, water is always the reference fluid; and 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 S_{wi} leading to a maximum relative permeability for oil of 1. If the absolute permeability is 100 mD with $K_{ro}(S_{wi}) = 0.9$, then the effective permeability is 90 mD. Which is equivalent to a base permeability of 90 mD and $K_{ro} = 1$. The product (90 mD) is also equal to the effective permeability.

Capillary pressure: P_c

This panel provides different options to implement the capillary pressure curves analytically or from existing data (Figure 36).

If the selected P_c 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 36).

P_c type

- The local P_c 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 P_c 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 P_c 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 61).

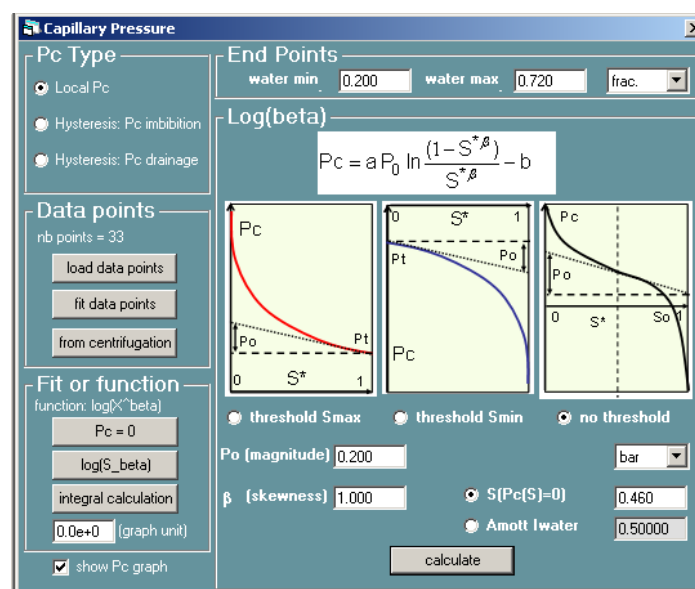


Figure 36: Capillary pressure window, with its side panel when Log(S_{beta}) has been checked.

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:

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

As “physical parameter”, the magnitude P_o and the saturation S_o are used which correspond to $P_c = 0$, related to b by:

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

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:

$$P_c \leq P_t \rightarrow S^* = 1 \quad ; \quad P_c > P_t \rightarrow P_c = P_o \frac{1}{S^*}$$

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

$$P_c \geq P_t \rightarrow S^* = 0 \quad ; \quad P_c < P_t \rightarrow P_c = -P_o \frac{1}{(1 - S^*)}$$

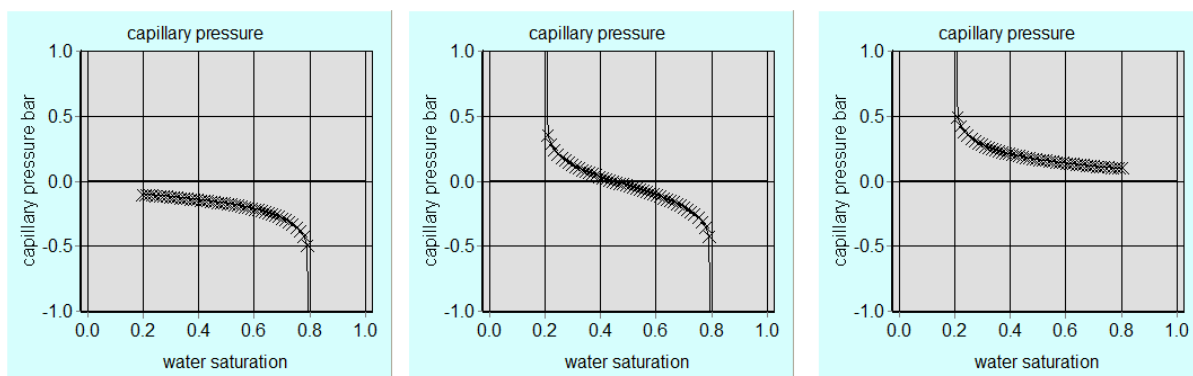
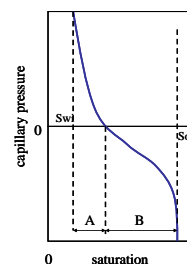


Figure 37: Examples of P_c curves obtained with the analytical Log(beta) function with 3 options: threshold S_{\min} ($P_t=100$ mbar); no threshold ($Sat=0.45$ at $P_c=0$), and threshold S_{\max} ($P_t=100$ mbar). For the three curves the magnitude is $P_o=100$ mbar.

Amott Index

Instead of fixing the point at which the P_c curve crosses $P_c(S) = 0$, 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 $I_o = B / (A + B)$.



Pc LET function (version 2018)

The Pc LET correlation function is described in: "A New Versatile Capillary Pressure Correlation, Lomeland et al., SCA, 2008, (Reference 18) and is usually used for Pc curves.

X described the normalized saturation: $X = \frac{X - X_{min}}{X_{max} - X_{min}}$, and the general form of the LET function is given by:

$$F_i(X) = \frac{X^{L_i}}{X^{L_i} + E_i(1 - X)^{T_i}}$$

with L, E, T three fitting parameters; the subscript i is defined by: $i \in \{forced, spont\}$

Three different cases are available: primary drainage, secondary imbibition, and secondary drainage.

Pc LET Equation for primary drainage:

$$P_c^{pd} = (P_{max} - P_t)F_{forced}(1 - X) + P_t, \text{ with } P_{max} > P_t \geq 0.$$

Pc LET Equation Secondary Imbibition:

$$P_c^{si} = (P_{max} - P_t)F_{spont}(1 - X) + (P_{min} - P_t)F_{forced}(X) + P_t, \text{ with } P_{max} \geq 0, P_{min} < P_t \leq 0$$

Pc LET Equation Secondary Drainage:

$$P_c^{sd} = (P_{max} - P_t)F_{forced}(1 - X) + (P_{min} - P_t)F_{spont}(X) + P_t, \text{ with } P_{max} > P_t \geq 0, P_{min} \leq 0$$

Relative permeabilities: Kr

The relative permeabilities control panel (Figure 38) 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 (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.

Relative permeability

Data Points

water: nb points = 18 ☒ Kr graph
oil: nb points = 18

Load data points Fit data points

JBN / J and R Kr steady-state

Fit or function

water Kr function: modified
oil Kr function: modified Power

Corey modified Corey LET

End Points

water **oil**

Kr Max 1.000 1.000

water saturation unit

Sat min 0.200 frac.

Sat max 0.675

Modified Corey

water **oil**

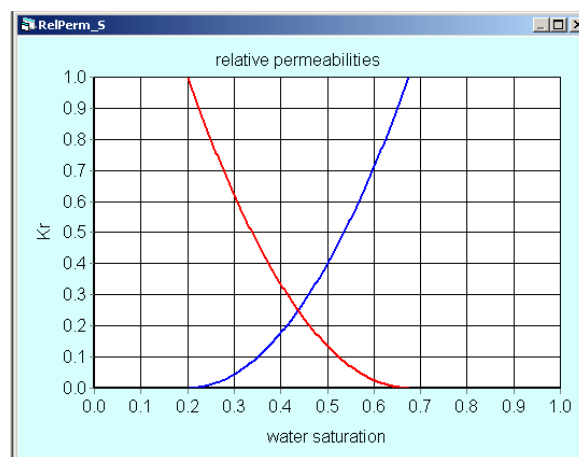


Figure 38: main window for relative permeabilities. Corey option, modified Corey, and Kr LET function with the corresponding kr curves.

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 38.

The Modified Corey function is defined as:

$$S^* = \frac{S_r - S_{\min}}{S_{\max} - S_{\min}} \quad ; \quad a = V - H - b \quad ; \quad b = 2\alpha(1 - H) - V - H$$

$$Kr_r = K_{r,\max} \left[\frac{a}{2\alpha} (S^*)^{2\alpha} + \frac{b}{\alpha} (S^*)^{\alpha} + (S^*)^H \right] \quad ; \quad Kr_n = K_{r,\max} \left[\frac{a}{2\alpha} (1 - S^*)^{2\alpha} + \frac{b}{\alpha} (1 - S^*)^{\alpha} + (1 - S^*)^H \right]$$

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.

Kr LET Function

The Kr 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

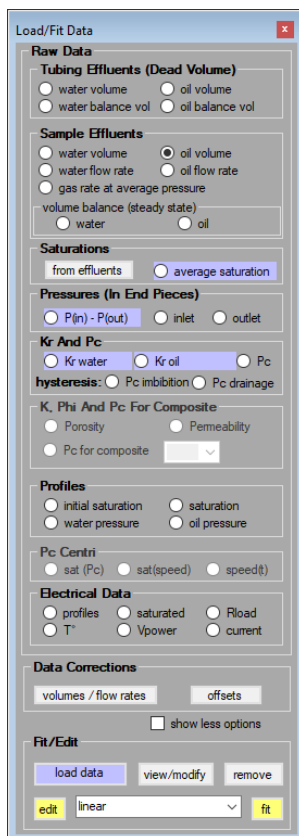


Figure 39: TPF Load/Fit data points window.

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 data editing section): interval, removing, adding or moving points, etc.

“**Fit**”: fit type specification.

The “**Show less options**” checkbox only displays essential and most used data, and hides advanced options.

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 incompressible, 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 2).

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 2: 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 used 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. 77 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. The liquid saturation is defined as the sum of the reference saturation and the saturation of the third immobile phase:

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

When loading data, S_{immo} can either be a constant value defined in the “**Type of Experiment**” window (Figure 40), 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

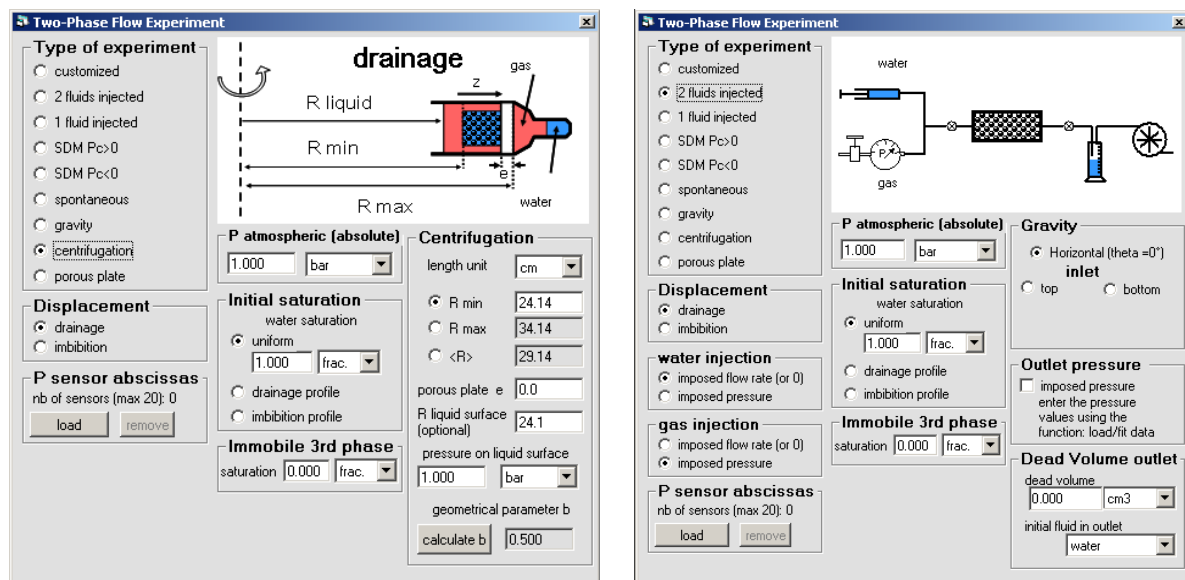


Figure 40: Type of experiment windows: a centrifuge case (left) and a steady-state case (right).

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”):

Initial saturation
ref saturation

☒ uniform
0.000 frac.

☐ drainage profile
☐ imbibition profile

- “**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

n	duration	rotation speed	inlet face Pc
1	0.500	500.000	0.38
2	0.500	500.000	0.38
3	5.000	1000.000	1.51
4	3.000	2000.000	6.03
5	2.000	3000.000	13.56

Figure 41: 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 3. 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	<table><tr><th>n</th><th>duration</th></tr><tr><td>1</td><td>0.000</td></tr></table>	n	duration	1	0.000				
n	duration								
1	0.000								
Porous plate	<table><tr><th>n</th><th>duration</th><th>Ref Fluid pressure</th><th>Nonref Fluid pressure</th></tr><tr><td>1</td><td>0.000</td><td>0.000</td><td>0.000</td></tr></table>	n	duration	Ref Fluid pressure	Nonref Fluid pressure	1	0.000	0.000	0.000
n	duration	Ref Fluid pressure	Nonref Fluid pressure						
1	0.000	0.000	0.000						
Kr Unsteady State	<table><tr><th>n</th><th>duration</th><th>Ref Fluid pressure</th></tr><tr><td>1</td><td>0.000</td><td>0.000</td></tr></table>	n	duration	Ref Fluid pressure	1	0.000	0.000		
n	duration	Ref Fluid pressure							
1	0.000	0.000							
Kr Steady State	<table><tr><th>n</th><th>duration</th><th>Ref Fluid rate</th><th>Nonref Fluid rate</th></tr><tr><td>1</td><td>0.000</td><td>0.000</td><td>0.000</td></tr></table>	n	duration	Ref Fluid rate	Nonref Fluid rate	1	0.000	0.000	0.000
n	duration	Ref Fluid rate	Nonref Fluid rate						
1	0.000	0.000	0.000						
Gravity	<table><tr><th>n</th><th>duration</th></tr><tr><td>1</td><td>0.000</td></tr></table>	n	duration	1	0.000				
n	duration								
1	0.000								
Centrifuge	<table><tr><th>n</th><th>duration</th><th>rotation speed</th></tr><tr><td>1</td><td>0.000</td><td>0.000</td></tr></table>	n	duration	rotation speed	1	0.000	0.000		
n	duration	rotation speed							
1	0.000	0.000							
Semi-Dynamic Pc>0	<table><tr><th>n</th><th>duration</th><th>Nonref Fluid rate</th></tr><tr><td>1</td><td>0.000</td><td>0.000</td></tr></table>	n	duration	Nonref Fluid rate	1	0.000	0.000		
n	duration	Nonref Fluid rate							
1	0.000	0.000							
Semi-Dynamic Pc<0	<table><tr><th>n</th><th>duration</th><th>Ref Fluid rate</th></tr><tr><td>1</td><td>0.000</td><td>0.000</td></tr></table>	n	duration	Ref Fluid rate	1	0.000	0.000		
n	duration	Ref Fluid rate							
1	0.000	0.000							
Customized									
<div><input checked="" type="radio"/> Nonref Fluid and Ref Fluid at constant rates (or 0)</div>	<table><tr><th>n</th><th>duration</th><th>Ref Fluid rate</th><th>Nonref Fluid rate</th></tr><tr><td>1</td><td>0.000</td><td>0.000</td><td>0.000</td></tr></table>	n	duration	Ref Fluid rate	Nonref Fluid rate	1	0.000	0.000	0.000
n	duration	Ref Fluid rate	Nonref Fluid rate						
1	0.000	0.000	0.000						
<div><input checked="" type="radio"/> Ref Fluid at constant pressure - Nonref Fluid at cst rate (or 0)</div>	<table><tr><th>n</th><th>duration</th><th>Ref Fluid pressure</th><th>Nonref Fluid rate</th></tr><tr><td>1</td><td>0.000</td><td>0.000</td><td>0.000</td></tr></table>	n	duration	Ref Fluid pressure	Nonref Fluid rate	1	0.000	0.000	0.000
n	duration	Ref Fluid pressure	Nonref Fluid rate						
1	0.000	0.000	0.000						
<div><input checked="" type="radio"/> Nonref Fluid at constant pressure - Ref Fluid at constant rate (or 0)</div>	<table><tr><th>n</th><th>duration</th><th>Nonref Fluid pressure</th><th>Ref Fluid rate</th></tr><tr><td>1</td><td>0.000</td><td>0.000</td><td>0.000</td></tr></table>	n	duration	Nonref Fluid pressure	Ref Fluid rate	1	0.000	0.000	0.000
n	duration	Nonref Fluid pressure	Ref Fluid rate						
1	0.000	0.000	0.000						

Table 3: 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: $Nb = \frac{K \langle r \rangle \omega^2 \Delta \rho}{\sigma \cos \theta}$, with K the absolute permeability, $\langle r \rangle$ the distance from the axis of centrifuge to the middle of the sample, ω the centrifuge rotation speed, and $\Delta \rho$ 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 v_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 P_c 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.

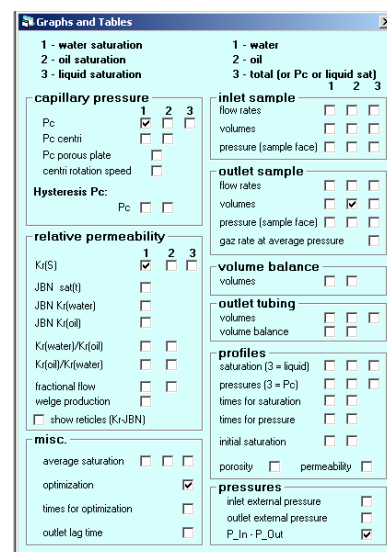


Figure 42: The “view” menu for selecting the graphs.

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 50) and are referenced in Cydar as “P sensor abscissas”. Figure 43 shows the P sensor abscissas frame in the window “type of experiment”.




Figure 43: 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.

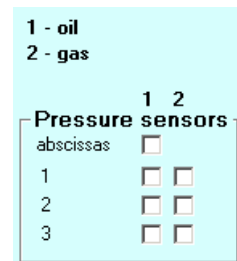
The  is a warning displayed on a tooltip over it. This warning indicates that the P sensors abscissas are different that the abscissas loaded with pressure profiles. Then only simulation curves will be shown on the $P(t)$ graphs.

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 44)

Figure 44: 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 47).

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 45).

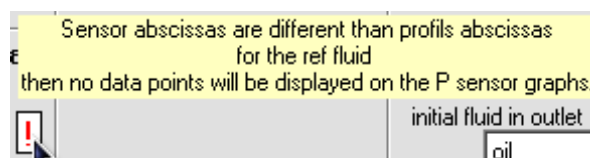


Figure 45: warning on the “P sensor abscissas” frame.

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 46).

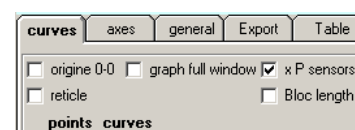


Figure 46: checkbox to display the abscissas of the pressure sensors on profile data.

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 47).

\	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 47: 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 $i+1$:

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

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

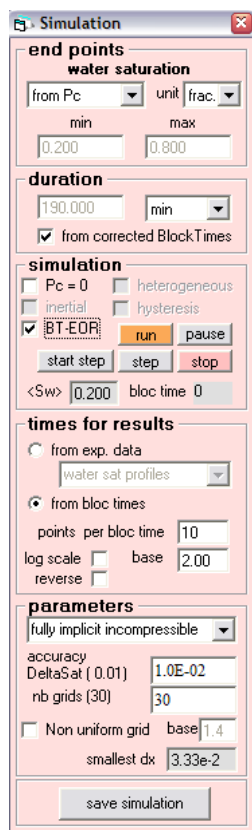


Figure 48: The simulation window of the two-phase flow module.

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.
- **"<Sw>"** 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.

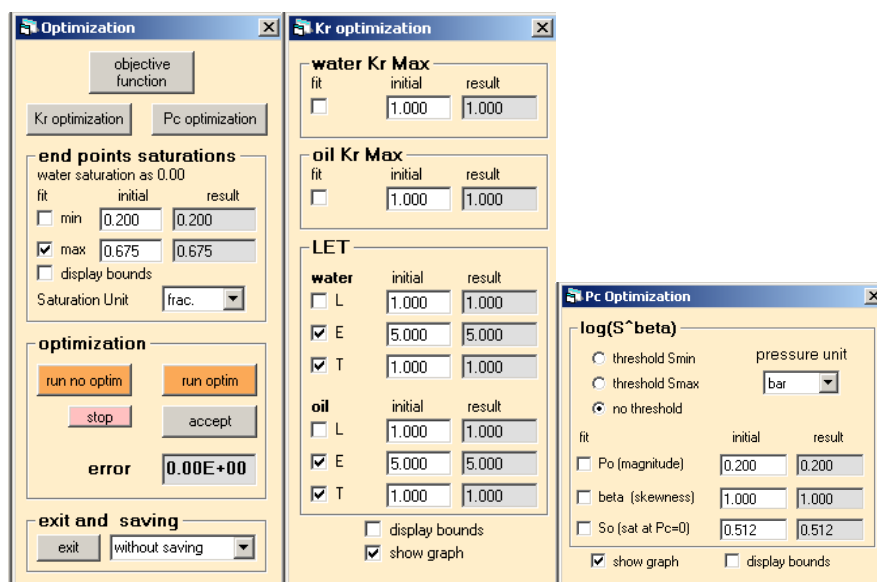


Figure 49: Optimization panels for Kr fitted with the Kr LET function and Pc with a function Log(beta).

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 Kr LET function for Kr; Log(beta) or Pc LET 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 49) 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 50).

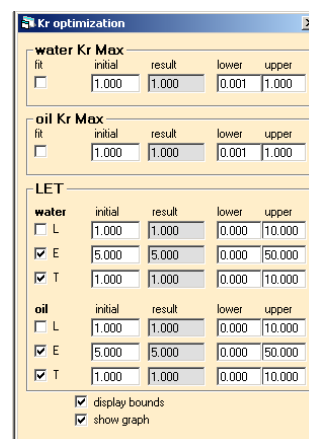


Figure 50: displaying bounds.

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 51, the user can impose different weights.

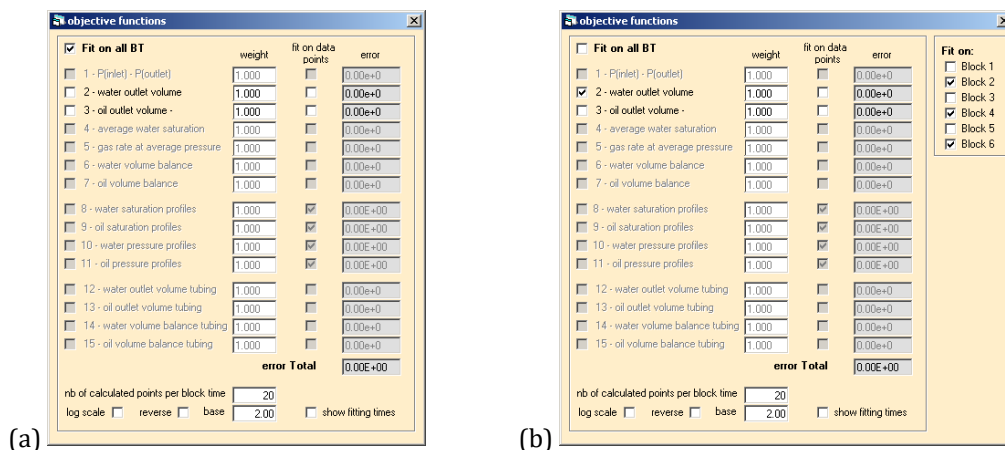


Figure 51: Define the objective function by selecting experimental data to be compared to results of numerical simulation.

The optimization can be done on all block times (as shown top left Figure 51a) or on selected block times (Figure 51b).

The user can choose the number of calculated points per block time and their position on linear or log scale (bottom Figure 51a). These latter 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 51a).

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 52).

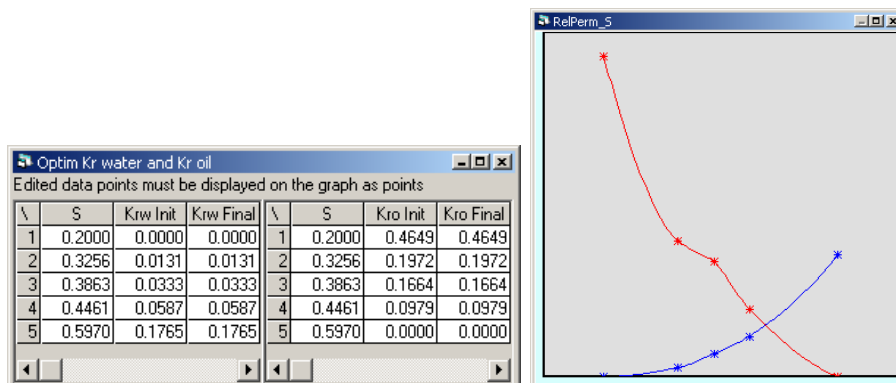


Figure 52: Interpolation knots table and the corresponding Kr graph.

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 53).
 - 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.

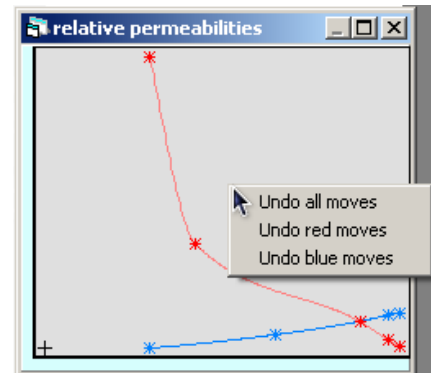


Figure 53: Right click popup in an optimization per points.

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.

OptimKr water and Kr oil							
\	S	Krw Init	Krw Final	\	S	Kro Init	Kro Final
1	0.2000	0.0000	0.0000	1	0.2000	1.0000	1.0000
2	0.4383	0.0310	0.0310	2	0.4383	0.2250	0.2250
3	0.6006	0.0622	0.0622	3	0.6006	0.0606	0.0606
4	0.6541	0.0767	0.0767	4	0.6541	0.0181	0.0181
5	0.6750	0.0822	0.0822	5	0.6750	0.0056	0.0056

Figure 54: 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.

OptimKr water and Kr oil							
\	S	Krw Init	Krw Final	\	S	Kro Init	Kro Final
1	0.2	0.0000	0.0000	1	0.2000	1.0000	1.0000
2	0.4383	0.0310	0.0310	2	0.4383	0.2250	0.2250
3	0.6006	0.0622	0.0622	3	0.6006	0.0606	0.0606
4	0.6541	0.0767	0.0767	4	0.6541	0.0181	0.0181
5	0.675	0.0822	0.0822	5	0.6750	0.0056	0.0056

Figure 55: 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 34), and the “**JBN-Jones and Roszelle**” window can be open (Figure 56).

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.

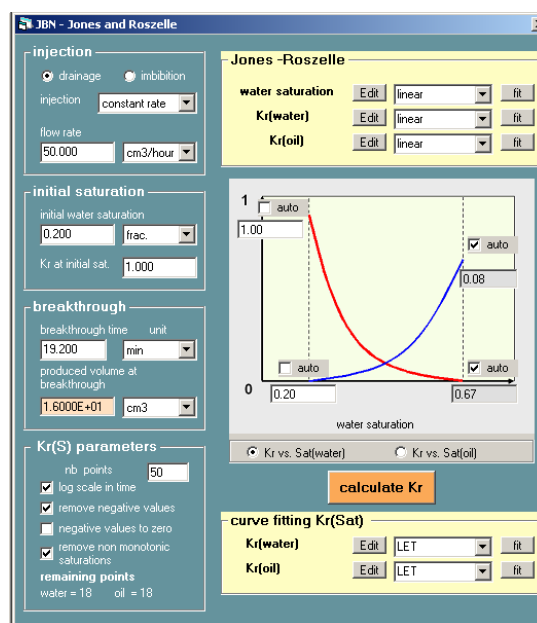


Figure 56: Window for JBN calculation.

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 K_r 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 K_r can correspond to the same value of saturation (Figure 57). This issue can be avoided by using the Jones and Roszelle method, which smooth the saturation and K_r as a function of time before eliminating the time variable.

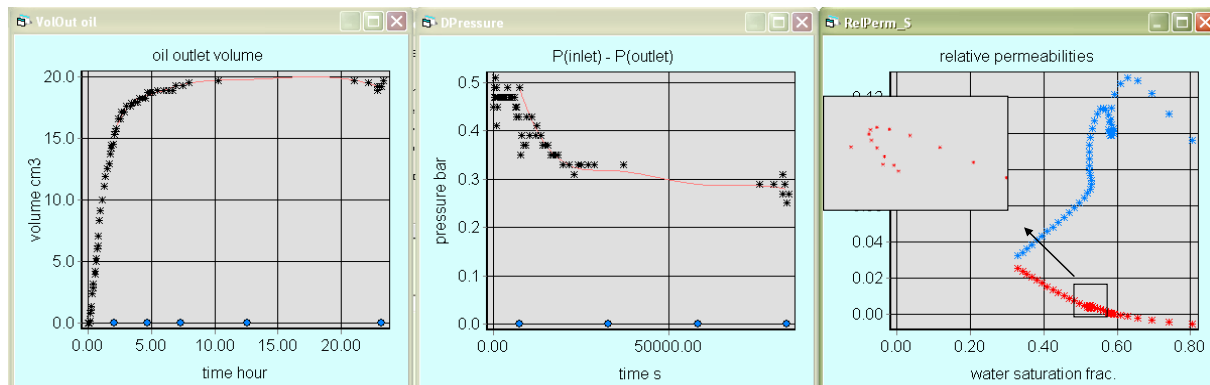


Figure 57: Example of K_r determination with noisy data leading to several K_r values.

Tutorial K_r _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 K_r USS:

- **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 P_c or K_r .
- **Tutorial_Kr_USS_MW_JBN.cyd**: analytical calculation of K_r done from experimental data, K_r fitted with power law (Corey) for optimization, and analytical P_c .
- **Tutorial_Kr_USS_MW_optim_initial.cyd**. This file is the starting point for numerical optimization. For this purpose, the JBN K_r 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 “ K_r 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 $S_w=0.679$.

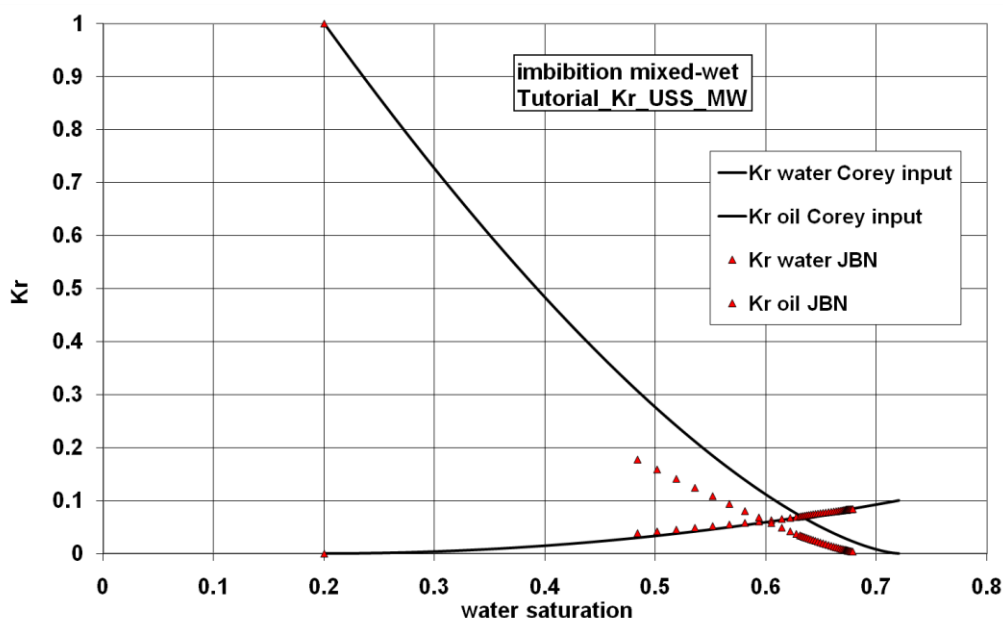


Figure 58: Example of Tutorial_Kr_USS_MW, input Kr (exact) and results of JBN calculation.

sat profiles

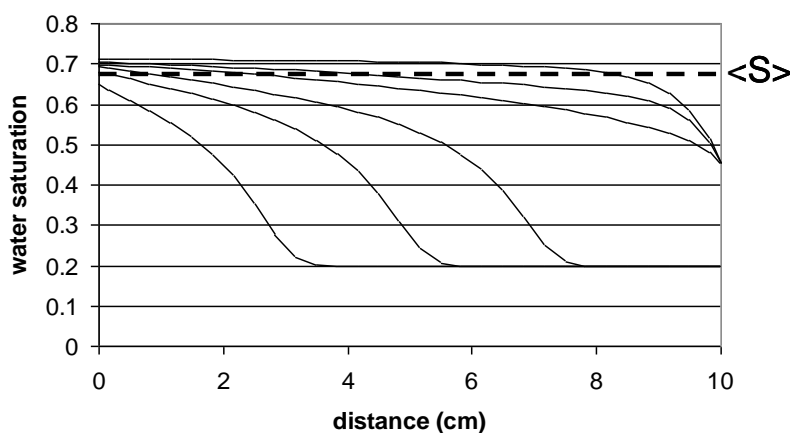


Figure 59: Example of Tutorial_Kr_USS_MW. The oil retention at the outlet is due to the negative part of the capillary pressure curve.

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.

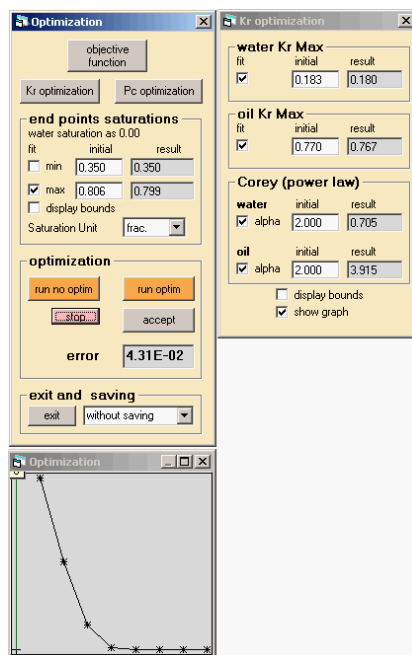
By default, the data will be linearly interpolated, but a Hermite interpolation gives a smoother curve. If a simulation is run with the linear interpolation, bumps due to the derivative change at the knots might be observed.

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.

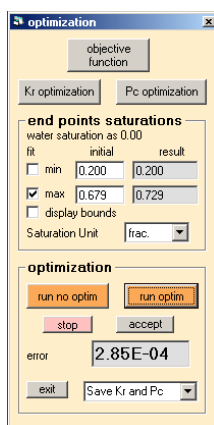


- Opening the "optimization" window: button "**optimization**";
- Selecting the data to fit and the "block times" on which to perform optimization: button "**objective function**", here keep "fit on all BT" and select data 1 and 3;
- Select Smax;
- Selecting Kr parameters: button "**Kr optimization**",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

To adjust this parameter automatically, check the fitting box and select delta pressure in the objective function. Then “**run optim**”. The fit gives $Kr_{max} = 0.1$.

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: $S_{max}=0.72$, $Kr_{max} \text{ water} = 0.10$, $n_{\text{water}} = 2.01$, $Kr_{max} \text{ oil} = 1.00$ and $n_{\text{oil}} = 1.4$.

Steady-state

Analytical

The use of this module requires using the average saturation. 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).

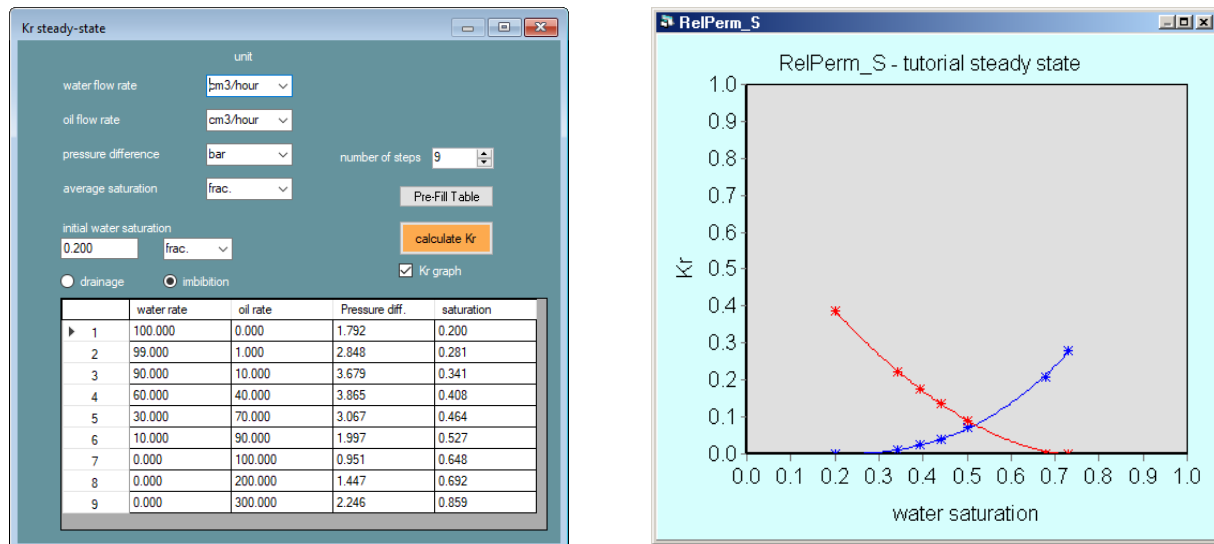


Figure 60: Kr steady-state window and the corresponding Kr result.

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.

The number of lines in the table can be adjusted from the “number of steps” text box, or by right-clicking on one line in the table.

The “Pre-Fill Table” function gathers information from the blocktimes, average saturation data, and pressure data and pre-fill the table. Rates are taken from block times. Pressure difference and saturation are calculated as the average of the 10 last points of each blocktimes.

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”:

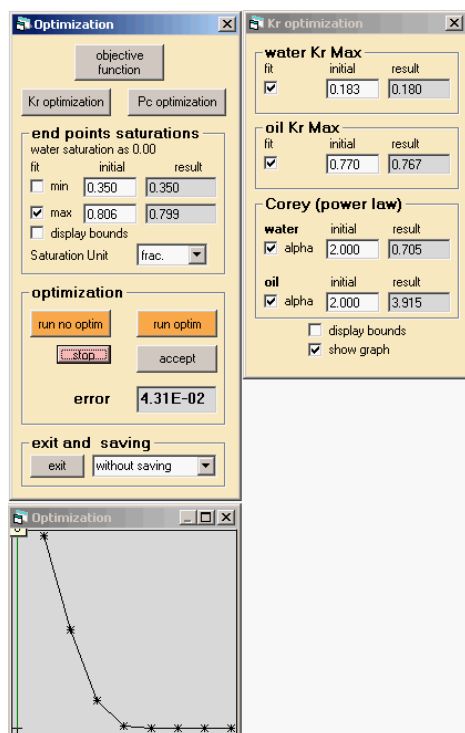
- Open the “capillary pressure” window: button “**Pc input**”;
- Select the analytical function “**Log(S_beta)**”: end points 0.2-0.8, “threshold Smin”, Po=0.1 bar, Pt=-0.1 bar.
- Press “**calculate**”

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 ($S_{\text{final}}=0.8$, $K_{r\text{max water}}=0.4$, $n_{\text{water}}=2.5$ and $n_{\text{oil}}=2$).



- 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 61)):

- “**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 $P_c(S)$, such as the calculated average curve $P_c(<S>)$ fits better the experimental average data. The Forbes continuous calculation is used as a first guess.

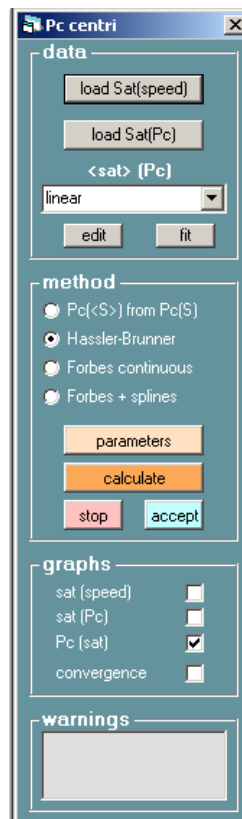


Figure 61: Pc centrifuge.

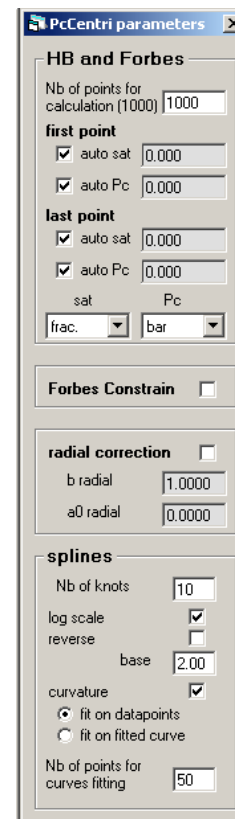


Figure 62: Parameters for the Pc centrifuge calculation

The user can control several parameters (Figure 62).

- 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.

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, $P_c(S)$ refers to local capillary pressure curve and $P_c(\langle S \rangle)$ 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 P_c curve from experimental $P_c(\langle S \rangle)$ curve.

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

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

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.

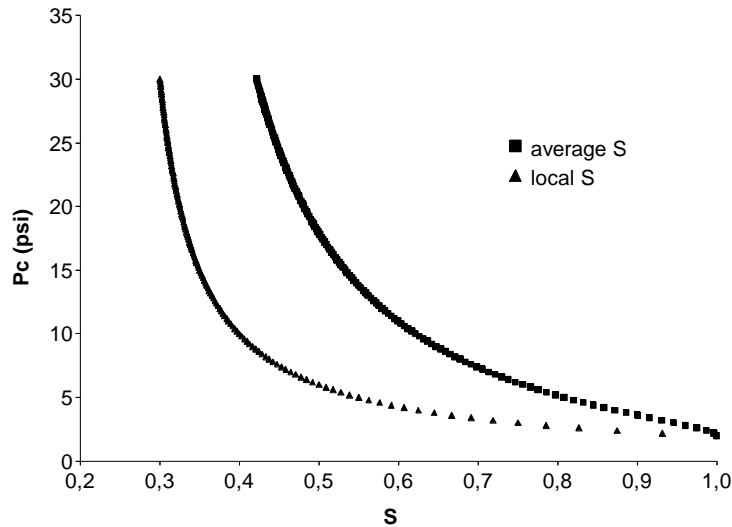


Figure 63: Theoretical $P_c(S)$ and $P_c(\langle S \rangle)$ curves, analytical case from Chen and Ruth (1993)

Figure 63 shows the analytical $P_c(S)$ and $P_c(\langle S \rangle)$ curves. We will now compare the results of the $P_c(S)$ curve calculation from the $P_c(\langle S \rangle)$ 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 $P_c(\langle S \rangle)$ curve has 20 points, calculation (and so derivatives) will be done with only 20 points. If the $P_c(\langle S \rangle)$ 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 $P_c(\langle S \rangle)$ curve (re-calculated from the $P_c(S)$ curve) and the experimental $P_c(\langle S \rangle)$ 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.

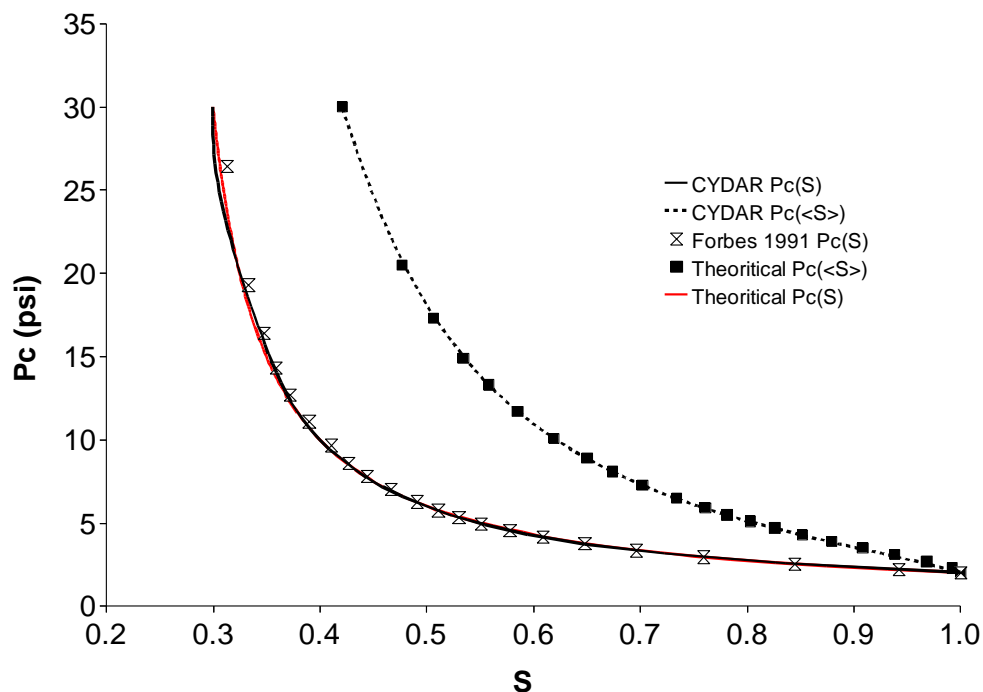


Figure 64: Analytical case with 23 points, comparison with Forbes SCA 1991 and CYDAR optimization tool.

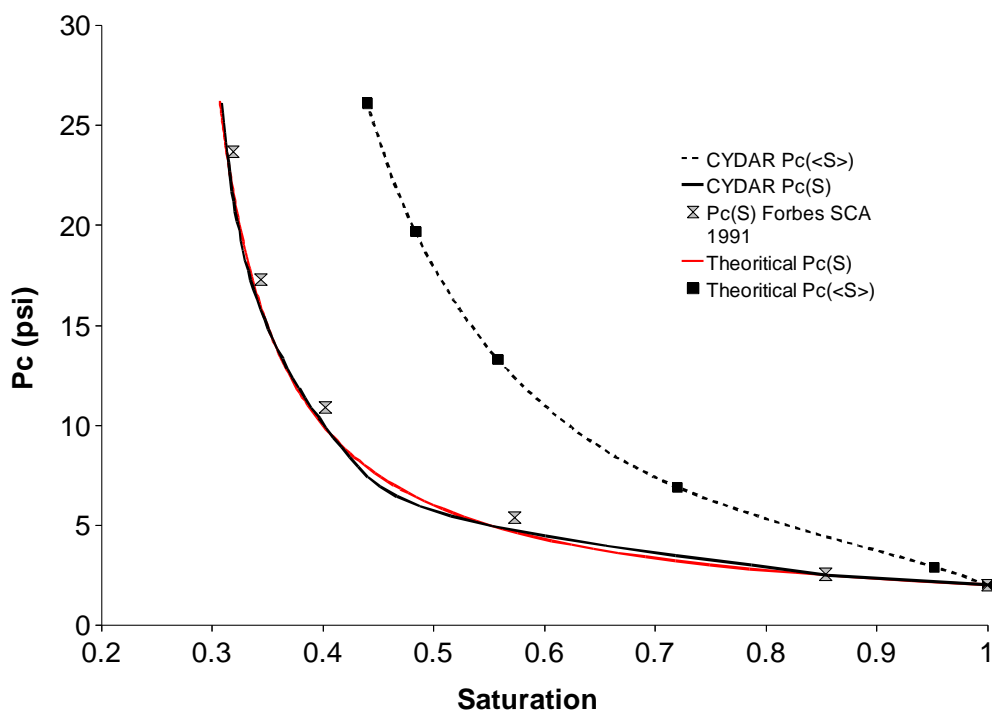


Figure 65: Analytical case with 6 points, comparison with Forbes SCA 1991 and CYDAR optimization tool.

Figure 64 shows results using a data of 23 points as original experimental data. Both methods give good results. The fits are good with the analytical $P_c(S)$ curve (red line Figure 64) and with the $P_c(<S>)$ curve (black squares Figure 64). The re-calculated $P_c(<S>)$ curve in the SCA 1991 Forbes method will always fit exactly the original curve because it simply does a reverse calculation.

Figure 65 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 $P_c(S)$ 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 $P_c(S)$ curve, we run a simulation with Corey as K_r and the experimental conditions shown Figure 66. Experimental setup and K_r 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 P_c curve can be calculated. The corresponding CYDAR file is "Tutorial_PcCentri Drainage_Cydar_Optim.cyd."

We open the window "Pc Centri" (see above Figure 61):

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

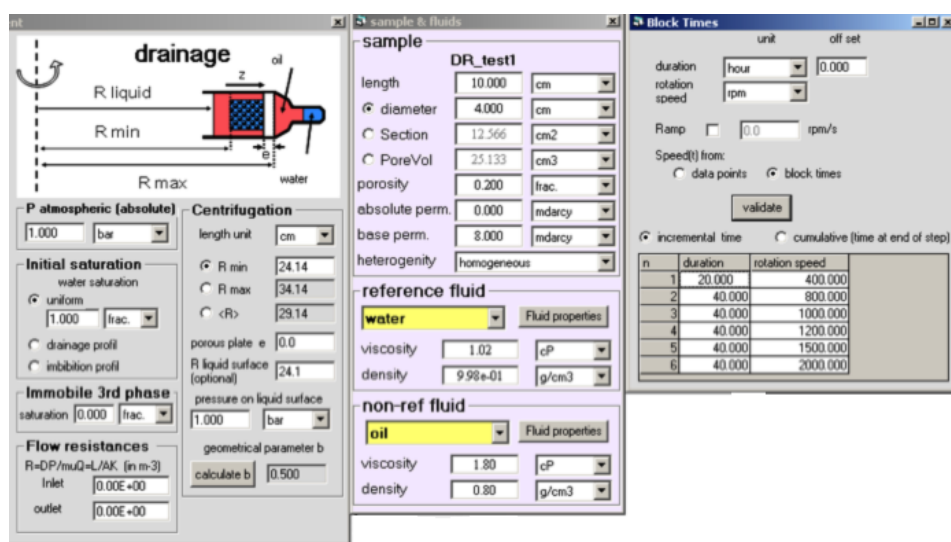


Figure 66: Simulation using analytical local saturation P_c curve.

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

Figure 67 shows the simulation results. In the graph the black stars are the experimental data (simulation using analytical P_c), the blue line is a simulation with the local P_c 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.

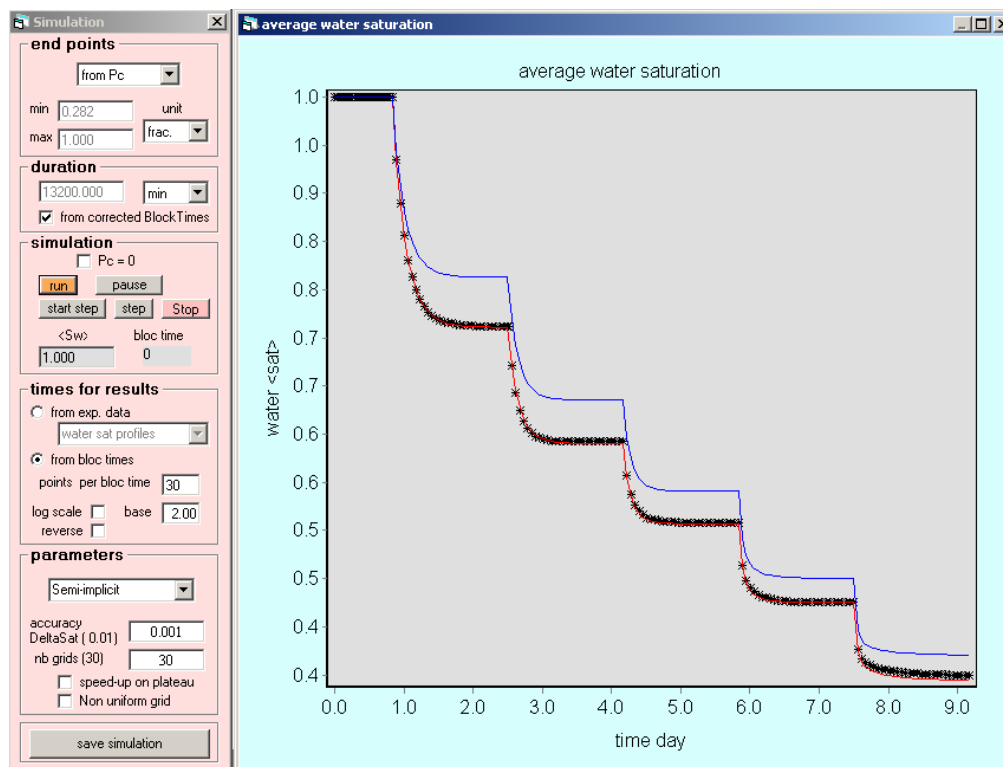


Figure 67: Simulation using the local P_c curve by both methods and comparison with the original data.

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

- file Tutorial_PcCentri_Drainage_Forbes_5_Pts.cyd;
- open the P_c centrifuge window (Figure 61) and choose the option " **$P_c(<S>)$ from $P_c(S)$** ";
- click "**calculate**";
- to display the calculated $P_c(<S>)$ versus the experimental data, open the $P_c(sat)$ graph and select curve "**edited points <sat>**" and "**calculated <sat>**".

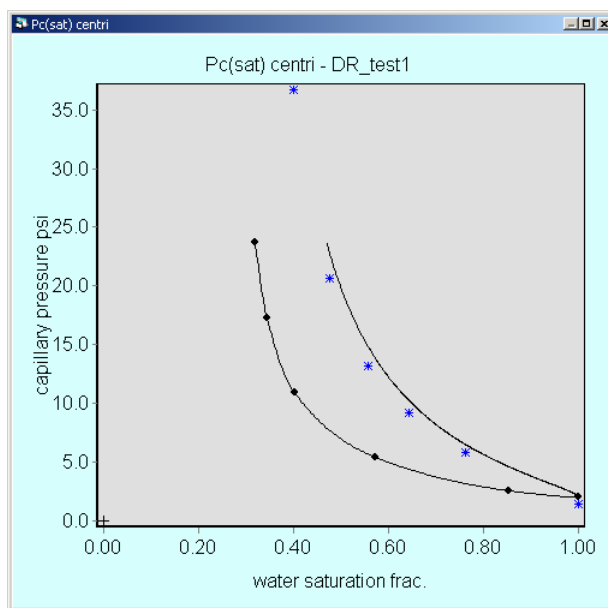


Figure 68: Comparison of the calculated and experimental $P_c(<S>)$, calculation done on the Forbes SCA 1991 local P_c curve.

Figure 68 shows that the calculated $P_c(<S>)$ from the local P_c 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

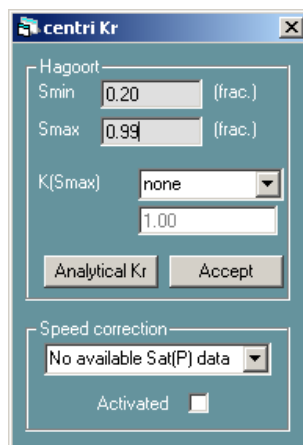


Figure 69: the "centri Kr" window, Hagoort part.

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

“Hagoort” frame:

- S_{min} and S_{max} are output. They give the calculated K_r 's end points.
- $K(S_{max})$ in drainage or $K(S_{min})$ in imbibition fix the extremum values.
- “**analytical Kr**” launches calculation.
- “**accept**” button loads calculation to data points. Any existing K_r data are erased.

Remark: The $K(S_{max})$ ($K(S_{min})$ 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 K_r curve can be fitted with the “**power**” fit function to obtain the Corey parameters (“**load/fit data**” frame, and select the corresponding K_r).

When displayed in log scale, the fit appears not optimal for lower values since the first point is taken as $K_r=0$ and corresponds to the asymptote (Figure 70). The saturation of the first point can be adjusted manually using the edit function (changing the X value), and pressing calculate. Figure 71 shows the result using a first point at $S=0.17$.

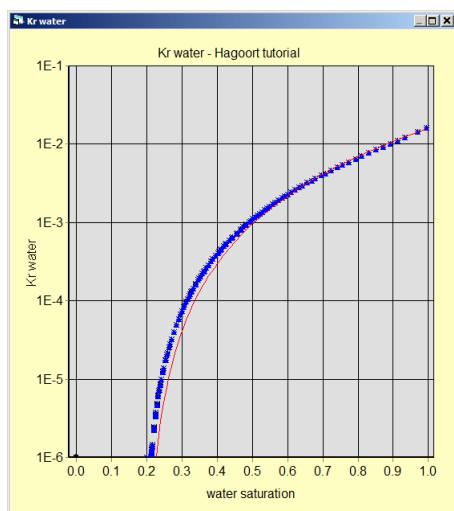


Figure 70: Corey fit of the tutorial, keeping the default value for the first point ($S=0.2$)

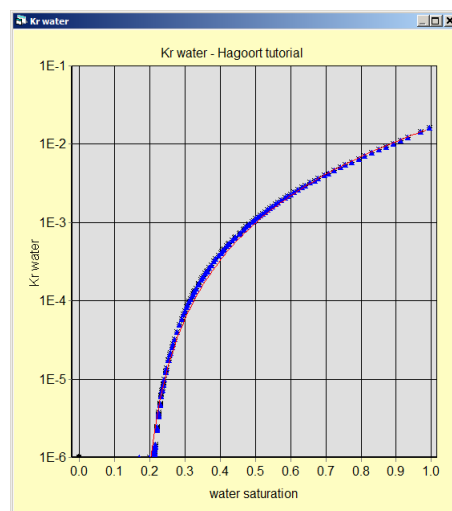


Figure 71 : Corey fit of the tutorial, adjusting the first point saturation at 0.17

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:

- $P_c=0$;
- Corey relative permeabilities: $K_{rw\ max}=0.016$ $n_{rw}=2.52$, $K_{rg}=1$ $n_{rg}=2$;
- End points: $S_{min}=0.2$, $S_{max}=1$.

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 72).

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

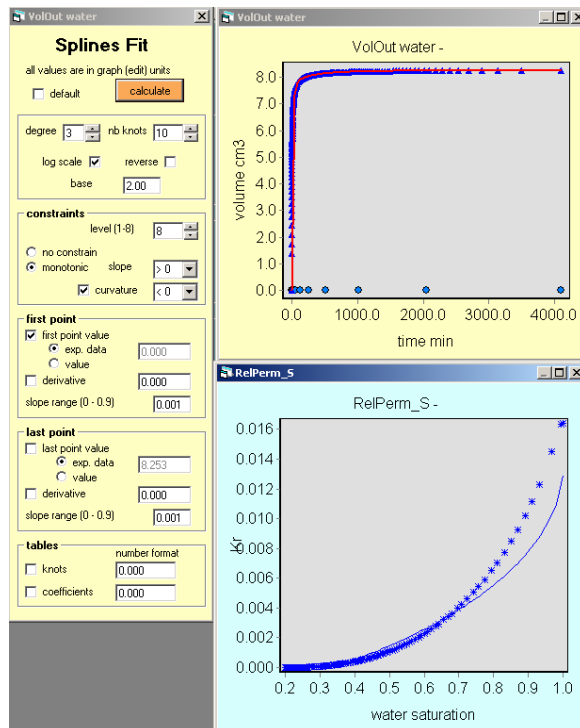


Figure 72: Hagoort calculation, effect of production data fit, splines with 10 knots.

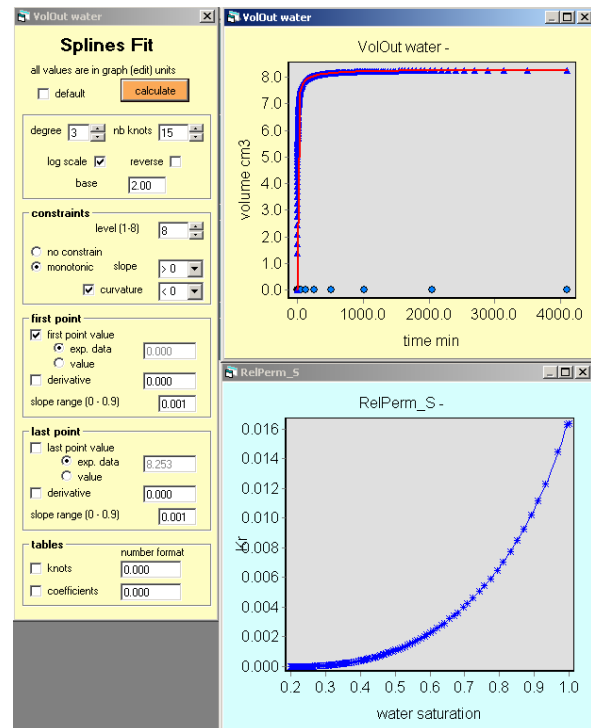


Figure 73: Hagoort calculation, effect of production data fit, splines with 15 knots.

Calculating Kr:

- Open the “**centri Kr**” window as explain above (Figure 69).
- 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 $K_{rw \max} = 0.016$ 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 72 and Figure

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 P_c curve.

Figure 74 shows an example of experimental average P_c curve shifted to fit the calculated curve. Experimental data points are red stars and the calculated curve is the black line.

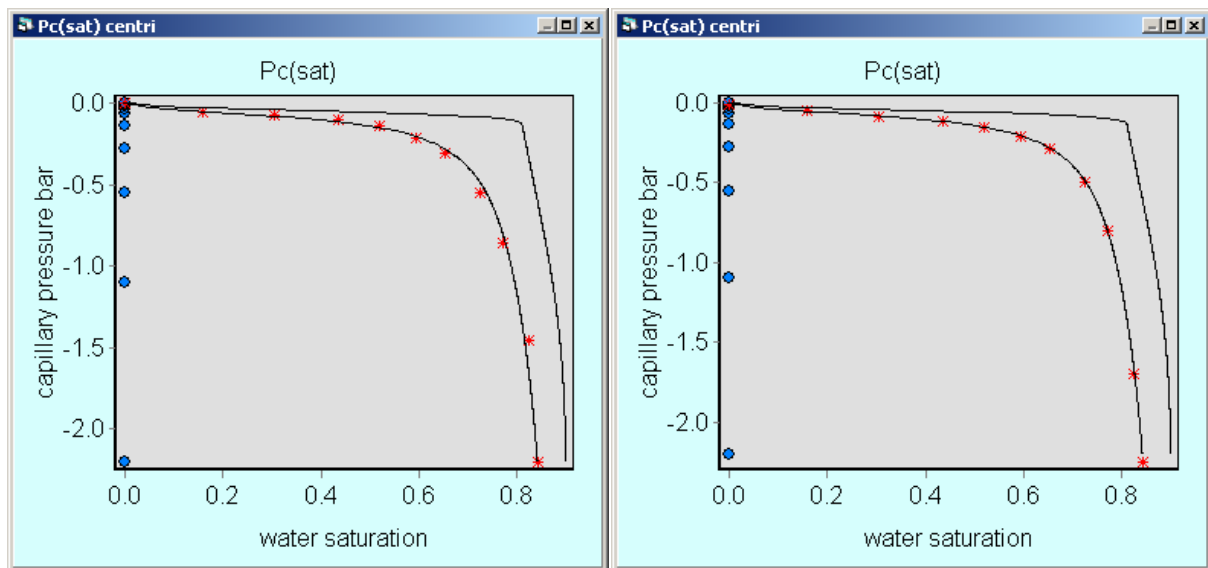
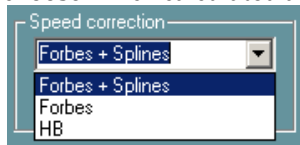


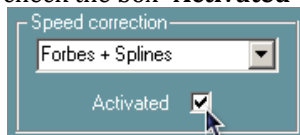
Figure 74: speed correction to fit calculated $P_c(<S>)$.

To activate speed correction:

- perform Local P_c calculation (see § Local P_c from average)
- open the Kr centri window (Figure 69);
- choose which calculated average P_c 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 74 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 through, 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 75). 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 75: 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 75). Table 4 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 4: 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.

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. Two options are available: either the same Pc curve for all blocks, or each blocks has his own Pc curve.

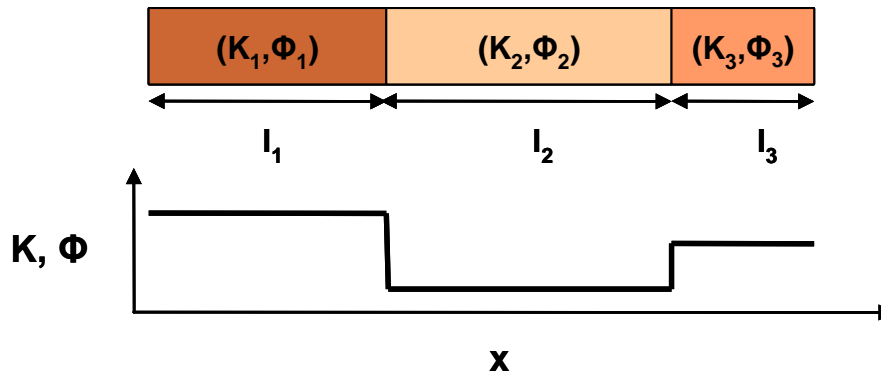
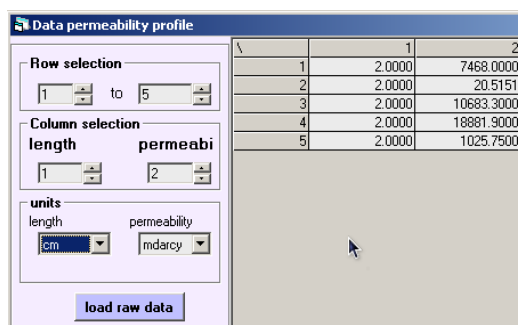


Figure 76: schematic composite sample.

Length, Permeability, and Porosity

First, the user loads data:



- Data are loaded in the “data points” window, in the “K, Phi and Pc for composite” frame.
- 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. 42). 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 77).

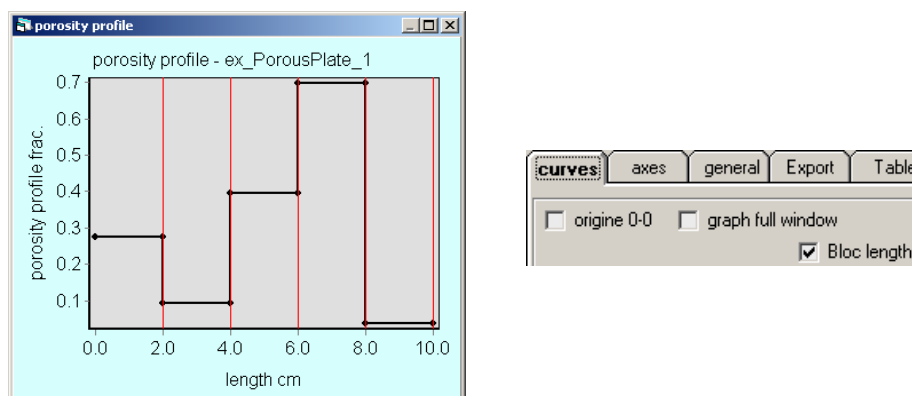


Figure 77: Displaying block lengths on porosity profile graph.

Pc curves

Composite with one Pc curve means a sample is made of several homogeneous blocks with only one Pc curve. The Pc curve is then entered manually in the “load data” window, in the “Kr and Pc” frame or generating a Pc curve in the “Pc Input” window.

Composite with multi Pc curve means a sample is made of several homogeneous blocks and each block has its own Pc curve. The Pc curves are then entered in the “load data” window, in the “K, Phi and Pc for composite” frame

Profiles

The permeability and the porosity are continuous properties versus length.

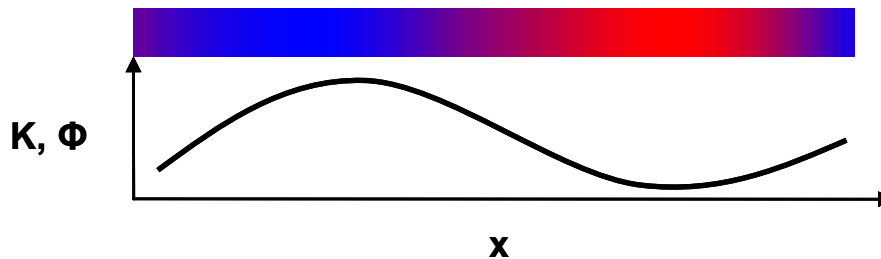


Figure 78: schematic sample characterized by porosity and permeability profiles.

First the user loads the data:

	1	2
1	1.0000	7468.0000
2	3.0000	20.5151
3	5.0000	10683.3000
4	7.0000	18881.9000
5	10.0000	1025.7500

- 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 79):

- “**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.

sample

length cm

☒ diameter cm

☐ Section cm²

☐ PoreVol cm³

porosity frac.

permeability mdarcy

heterogeneity

permeability and porosity from data.

permeability and porosity for capillary pressure

porosity frac.

permeability mdarcy

permeability and porosity for capillary pressure

porosity from average values

permeability user defined

reference fluid

block 2

block 3

block 4

block 5

a)

b) composite

c) profile

Figure 79: Sample properties: porosity and permeability input used for the Leverett calculation.

Simulation

simulation

☐ Pc = 0 ☐ heterogeneous

<Sw> bloc time

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.

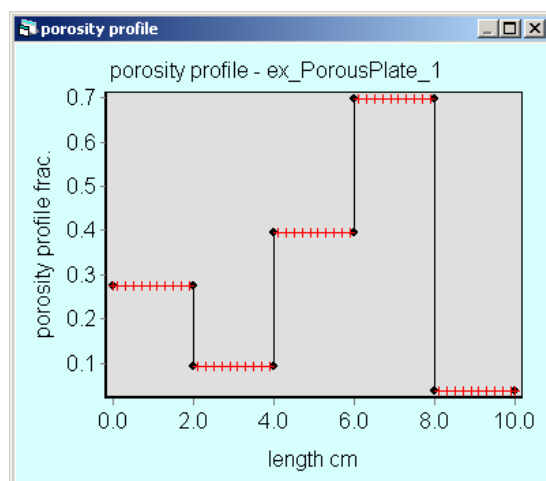


Figure 80: heterogeneity enabled and 10 grids per blocks.

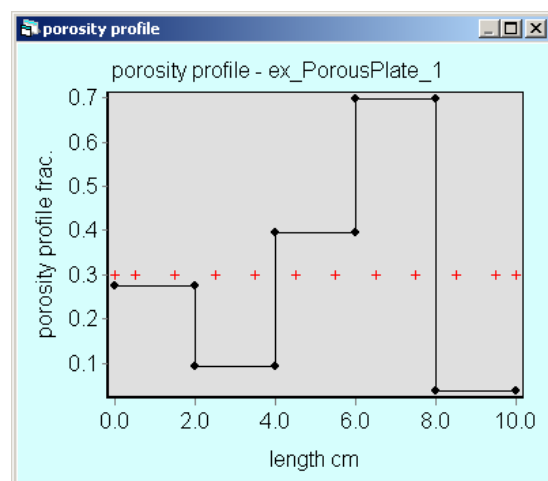


Figure 81: heterogeneity disabled and 10 grids.

Profiles

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

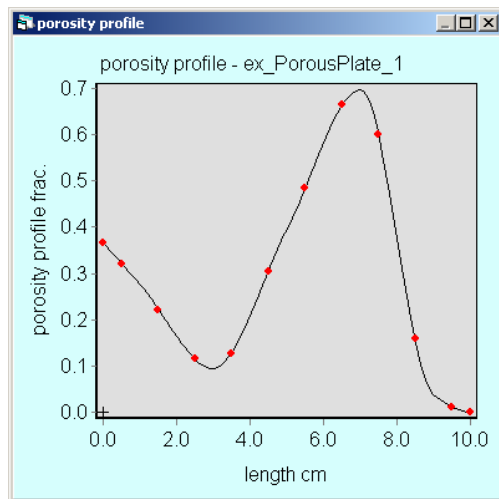


Figure 82: heterogeneity enabled and 10 grids.

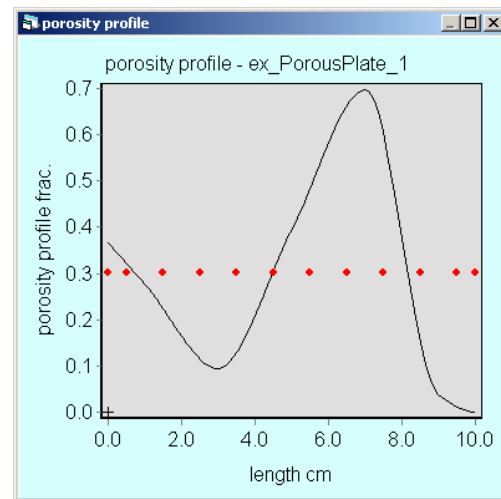


Figure 83: heterogeneity disabled and 10 grids.

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 84). We want to simulate an imbibition from this state, the sample is at S_{ini} on the drainage Pc curve P_c^D , but we only have an imbibition Pc curve P_c^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

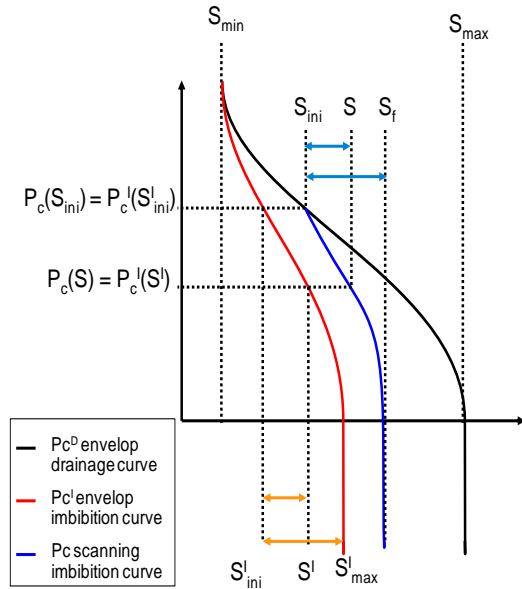


Figure 84: hysteresis, imbibition scanning curve generation.

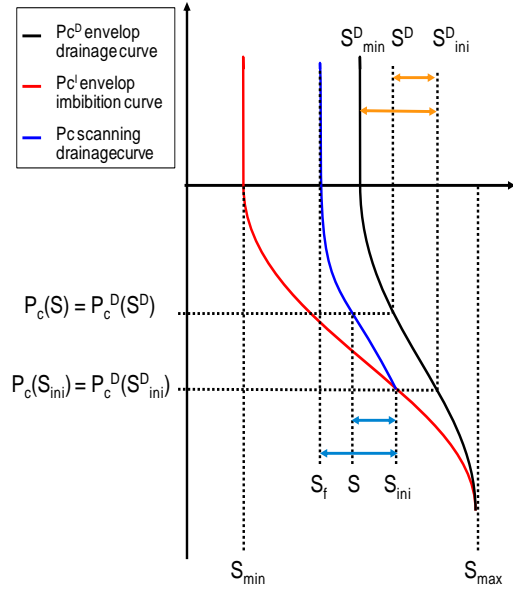


Figure 85: hysteresis, drainage scanning curve generation.

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

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

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

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

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

Pc scanning drainage curve

The principle is to generate a drainage scanning curve P_c starting at S_{ini} from the envelop imbibition curve P_c^I by calculating the saturation S of P_c such as $P_c(S) = P_c^D(S^D)$ (Figure 85) assuming:

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

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

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

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

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 86) meets the envelop curve at the origin of the first scanning curve (dark blue curve starting from black curve (Figure 86). For the moment in CYDAR the second scanning curve has the same final saturation calculation that if it is the first scanning curve.

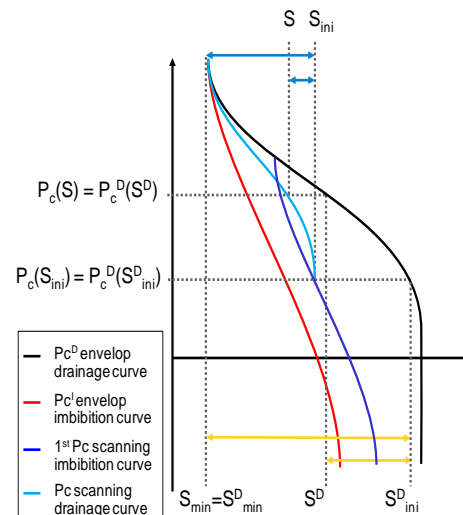


Figure 86 Hysteresis loop

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 23. 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 35). By default the entries are hidden and can be displayed by checking the “show inertial parameter” box (Figure 87).

<input checked="" type="checkbox"/> show inertial parameters			
	<input checked="" type="radio"/> B	<input type="radio"/> β	
water	0.00	0.00E+0	1/dm
gas	2.22	5.00E+5	1/dm

Figure 87: 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 23).

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 48). 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 Method (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

$$RI = S_w^{-n_1}$$

- "bending down" and "bending up" laws from Fleury¹⁶:

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

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

$C = 0$ 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

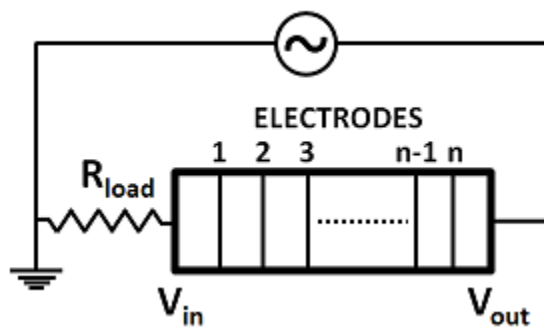


Figure 88: Rload at the inlet

The current is given by

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

Rload at the outlet

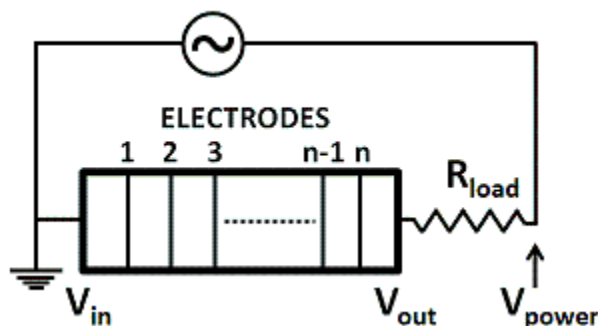


Figure 89: Rload at the outlet

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

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

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 Archie's law case:

$$S_{wi} = RI_i^{-\frac{1}{n_1}}$$

The average saturation is calculated by:

$$\langle S_w \rangle = \frac{\sum dx_i \Phi_i A S_i}{V_p}$$

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 90 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.

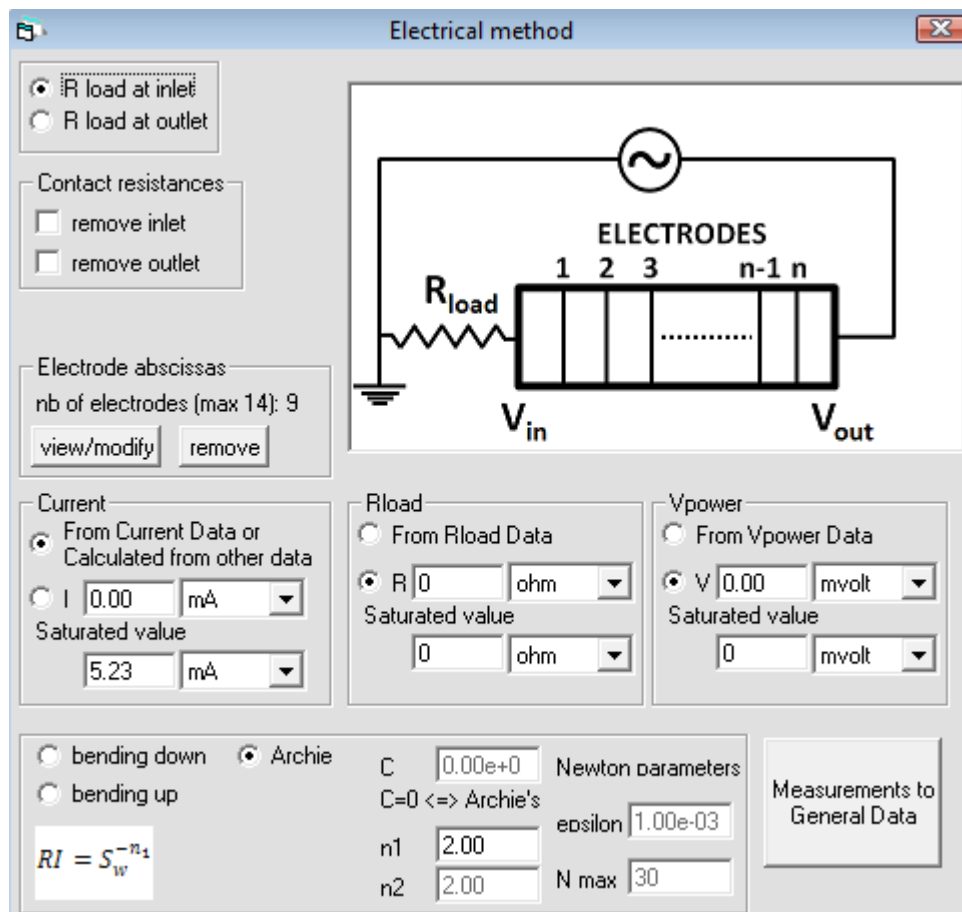


Figure 90: the "Electrical window" is the window with setup parameters

The experimental data are resistances:

The only parameters used in that case in Figure 90 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

The window contains the following elements:

- Radio buttons: ☐ bending down, ☒ bending up
- Radio button: ☐ Archie
- Field: C = 1.00e+0
- Text: Newton parameters
- Text: C=0 <=> Archie's
- Field: epsilon = 1.00e-03
- Field: n1 = 2.00
- Field: n2 = 2.00
- Field: N max = 30
- Formula: $RI = S_w^{-n_1} \frac{1 + CS_w^{-n_2}}{1 + C}$

Figure 91: 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 91 contains the different parameters. epsilon is the stopping criterion and Nmax is the maximum of iteration.

Input data

The window contains the following elements:

- Radio buttons: ☒ profiles, ☐ saturated, ☐ Rload, ☐ T*, ☐ Vpower, ☒ current

Figure 92: data related to electrical method in data points window

The image shows a software window titled 'Load Data'. It has two main sections. The top section is labeled 'units' and contains two dropdown menus: 'length' set to 'cm' and 'resistance' set to 'ohm'. The bottom section is labeled 'Electrical data' and contains a dropdown menu set to 'resistance', a text input field with '8', a dropdown menu set to 'ohm', and a checked checkbox labeled 'saturated'. Below these is a button labeled 'load profiles' and a text label 'number of profiles: 7'. At the very bottom, there is a section labeled 'times' with a 'row' label.

Figure 93: Part of Load Data 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 Load Data window (see Figure 93). 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 $\langle S \rangle$ 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 90.

The image shows a window titled 'Electrical'. It contains a list of checkboxes for selecting data to be graphed. The first checkbox is labeled 'abscissas' and is checked. Below it is a checkbox labeled '<S> sample' which is also checked. Then there are nine checkboxes numbered 1 through 9, all of which are unchecked.

Figure 94: Graphs relative to electrical data

TPF Reporting

Principle of report file

The report can be described as two parts: one with parameters, sample properties (Figure 95); and the second part with the data.

On Figure 95:

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:

- P_c and K_r 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: $P_c(<S>)$; forbes + splines if calculation have been done.

Client: Cydarex Experimentator: Lenormand Date of Experiment: 12/02/2006 Date of Interpretation: 12/02/2006 Series Reference: 0 Well: USS1 Location: rueil-malmaison Formation: Drilling Fluid: Rock Type: unknown Mineralogy: Depth Type: MDRT driller from: 0 to: 0 m		BLOCK 1	
SAMPLE name: heterogeneity type: homogeneous length: 10 cm diameter: 4 cm absolute perm.: 0 mdarcy porosity: 0.3 frac. base perm.: 100 mdarcy porevolume: 37.699 cm3		EXPERIMENT displacement: imbibition gravity: horizontal external pressure: 0 Pa initial saturation: 0.2 frac. third phase: 0 frac.	
		SIMULATION Sat min: 0.2 frac. Sat max: 0.72 frac. solveur: Fully implicit accuracy dS: 0.01	
		BLOCK 2	
WATER liquid type: brine viscosity: 1 cP density: 1 g/cm3 compressibility: 0 1/bar temperature: 20 Celsius pressure: 1 bar		OIL liquid type: brine viscosity: 10 cP density: 0.8 g/cm3 compressibility: 0 1/bar temperature: 20 Celsius pressure: 1 bar	
		BLOCK 3	
		BLOCK 4	
Pc ANALYTICAL analytical type: linear Sat min: 0.2 frac. Sat max: 0.72 frac.		Kr SIMULATION analytical type: Corey Sat min: 0.2 frac. Sat max: 0.679 frac. Kr max: 0.1 1 n: 1.877 1.544 H: V:	
		Kr BACKUP CURVE analytical type: Sat min: 0 frac. Sat max: 0 frac. Kr max: water oil n: H: V:	
		JBN Injection: constant rate rate: 50 cm3/hour breakthrough: 19.2 min	

Figure 95: First part of a report for an unsteady-state experiment.

Using template:

The main goal of the 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 96 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.

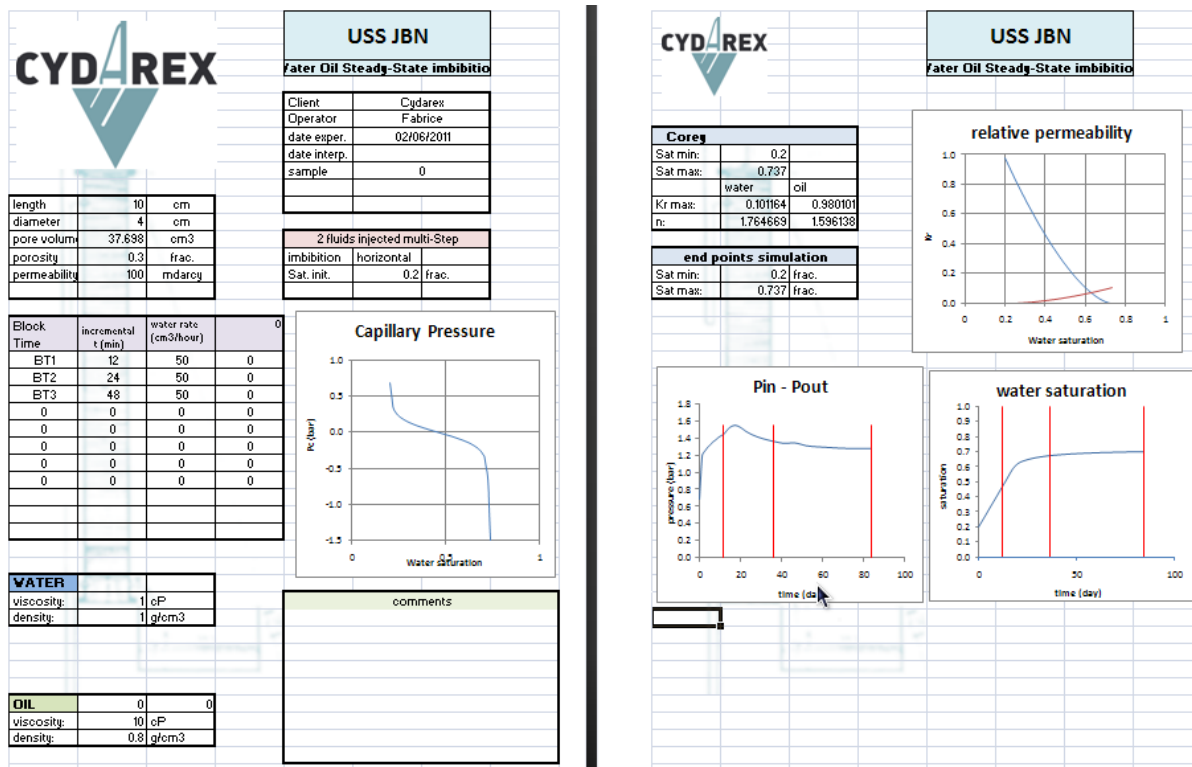



Figure 96: example of a more complex layout using template.

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 capillary 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, SCA 2003-31, 2003.
17. Lomeland F. and Ebeltoft E., A New Versatile Capillary Pressure  Correlation, SCA, 2008-08, 2008.