



Heriot-Watt University

SQUEEZE

Version 8.0, July 2013

USER'S MANUAL

Oscar Vazquez, Eric Mackay and Ken Sorbie

TABLE OF CONTENTS

1	INTRODUCTION TO SQUEEZE	4
1.1	Installation of SQUEEZE.....	5
1.2	Updating installed version of SQUEEZE	7
1.3	About this Manual	7
2	USER INTERFACE FOR DATA INPUT	7
2.1	Multiple Simulations	8
2.2	Simulation Type	9
2.3	Help Button	9
2.4	About Button	9
2.5	Isotherm Fitting Button	9
2.6	Project Title	11
2.7	Controls Functionality Grouping	11
2.7.1	Read File Button	11
2.7.2	Basic Parameters Button	11
-	Simulation Type	12
-	Model type.....	12
-	Units	13
-	Isotherm Derivation Parameters	13
-	Basic Model Parameters	14
-	Well Friction.....	14
-	Temperature Calculation Parameters.....	14
2.7.3	Edit Model.....	15
-	Layer properties.....	15
-	Component Properties	20
-	Stages Properties.....	31
-	Graphics Options	38
2.8	Run Model	39
2.9	Abort Model	39
2.10	Graphics functionality group.....	39
2.10.1	Output Frequency, (MOD).....	40
2.10.2	GRAPH CROSS.....	40

2.10.3	GRAPH TIME	40
2.10.4	GRAPH RAD.....	42
2.11	CLOSE ALL.....	43

1 INTRODUCTION TO SQUEEZE

SQUEEZE is a software package for modelling oilfield scale inhibitor squeeze treatments both in laboratory core floods and in the field. SQUEEZE offers significant enhancements to the earlier versions, it provides a wider suite of tools for modelling both field and experimental scale inhibitor squeeze treatments. A wide range of options is available in SQUEEZE, where all the previous model are retained in this last version, including Hong and Shuler analytical model, the Isotherm validation and derivation and the 1D coreflood and radial near-well model, although they have been improved in various ways. The most important new facilities and modifications that have been implemented in SQUEEZE are as follows:

- (i) **Two phase flow**, the simulation of water and oil phase is included, in particular an aqueous and a non-aqueous phase.
- (ii) **Automatic Isotherm History Matching**, Isotherm history matching for a given field return profile may require effort and time, this tool provides an automatic method.
- (iii) **The Data-Input Interface** has been completely re-written in Visual C++ for modifying project input data files. This offers a much more flexible, user-friendly and robust data processing tool.
- (iv) **The Graphical Output** for displaying real-time simulation results has been completely rewritten. This offers a much more flexible, user friendly and robust processing tool.

The flowchart shown in Figure 1.1 summarises the SQUEEZE models and their functions. The details of operating each specific branch of the program will be introduced in the relevant chapters of this manual.

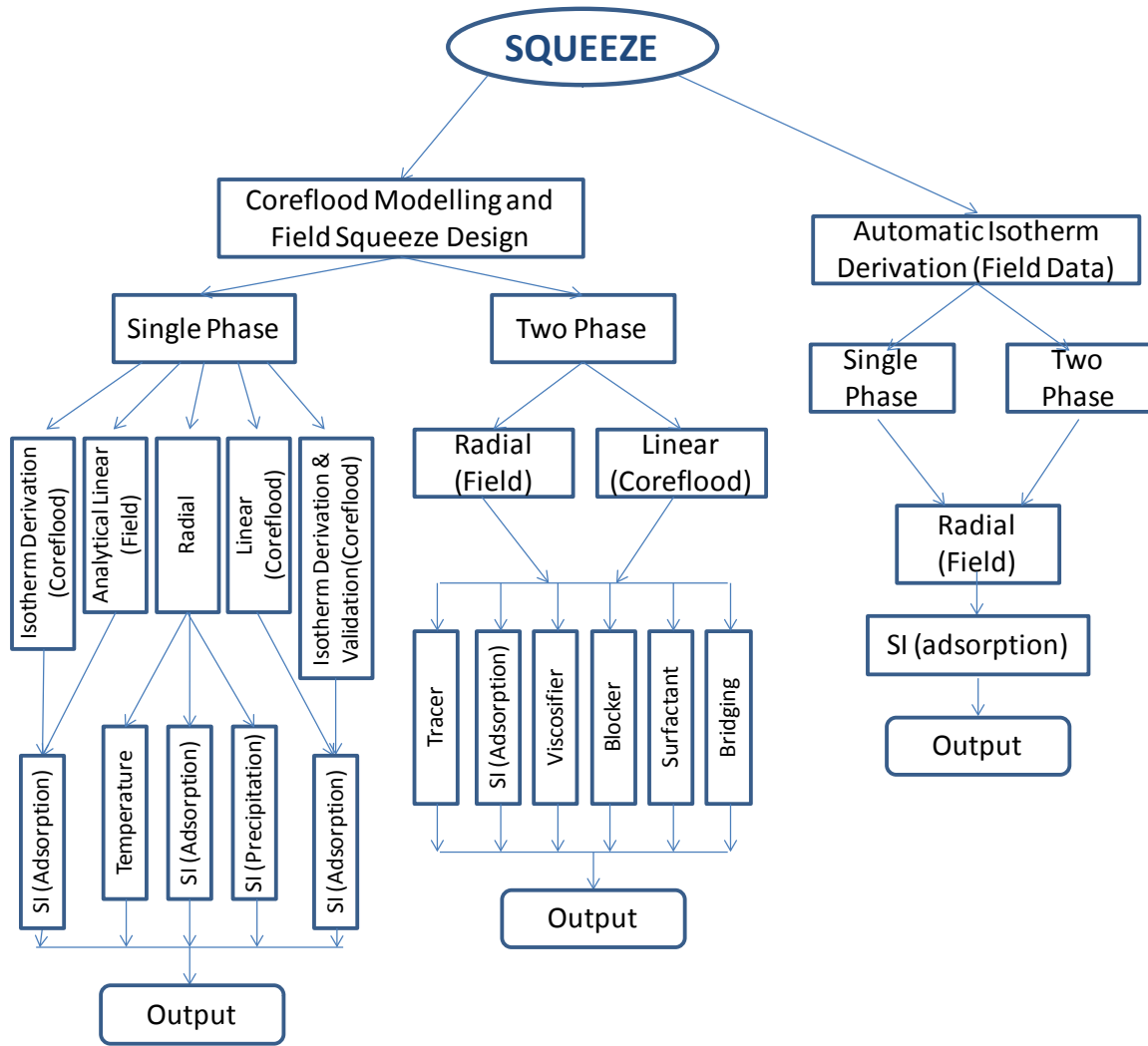


Figure 1.1 SQUEEZE model overview.

1.1 Installation of SQUEEZE

Download the zip file SQUEEZE8_full.zip from dscale.org; this file contains the folder and subfolders necessary to run SQUEEZE. When you extract the files from this zip file to a location of your choice, you should have the folder/subfolder tree structure shown in the illustration below. For SQUEEZE to function correctly you must ensure that the subfolders are located in the same parent folder as the SQUEEZE executable.

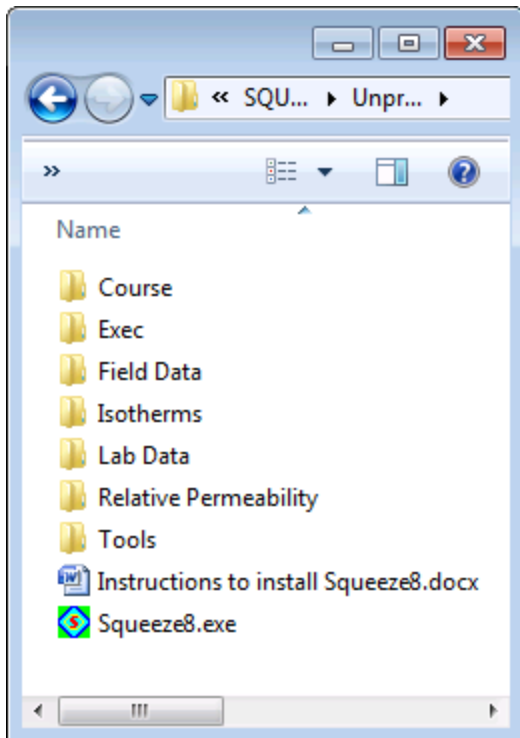


Figure 1.2 SQUEEZE folder/subfolder tree structures.

A description of the contents and use of each folder can be found below:

- i. **Course:** This folder contains the materials for the beginners and advanced courses, including example files, presentations, exercises, useful tools and a selection of papers about squeeze treatment modelling.
- ii. **Field Data*:** The field return profiles from scale inhibitor squeeze treatments are stored in this folder, which also includes a template “FieldData-Template.txt”.
- iii. **Exec*:** This folder contains additional essential executables.
- iv. **Isotherms*:** This folder is used to store isotherm tables, which are generally derived from coreflood experiments; a template file “Isotherm-Template.txt” is included in this folder.
- v. **Lab Data*:** This is where SI effluent concentration profiles from laboratory corefloods tests are stored; a template file “LabData-Template.txt” is to be found in this folder.
- vi. **Relative Permeability*:** Oil and water relative permeability curves in table format are stored in this folder, which includes a template file “RelPerm-Template.txt”.
- vii. **Tools:** Various tools may be found in this folder, including the necessary files to install the HASP device driver.

***Note:** All subfolders with an asterisk are required to allow SQUEEZE to function properly. They must be located in the same parent folder as the SQUEEZE executable.

1.2 Updating installed version of SQUEEZE

If you have already installed a version of SQUEEZE and would like to install a newer version, it should just be a case of downloading the executables, although depending on how old your last version was, you may also need to install the relevant driver for the HASP key. The following steps should be carried out:

- i. Replace the executable file “Squeeze8.exe” in the same location.
- ii. Replace contents of the folder “Exec”.

1.3 About this Manual

In addition to a general discussion of the functions and capabilities of the SQUEEZE models, this manual is intended to be a training manual which aims to take you through the dialogueues to build your own model input data files. SQUEEZE input data has been designed as a number of intuitive dialogueue based modules, where a number of models can be included in the simulation project. Emphasis is placed on the actual physical/chemical problem which is being modelled rather than on a description of the mathematical and numerical details of SQUEEZE.

For further assistance and/or support, please contact:

Oscar Vazquez	Oscar.Vazquez@pet.hw.ac.uk	+44 131 451 3609
Eric Mackay	Eric.Mackay@pet.hw.ac.uk	+44 131 451 3670
Heather O’Hara	Heather.OHara@pet.hw.ac.uk	+44 131 451 4312

2 USER INTERFACE FOR DATA INPUT

The SQUEEZE interface is an interactive program for creating; editing and/or modifying SQUEEZE input data files. The interface employs standard Microsoft dialogue windows, see Figure 2.1. This is the main dialogue which gives the user full access to the all the models to build any project, as well as editing any project, in addition it gives full access to the graphical output. The functionalities are divided in different groups or are standalone functionality.

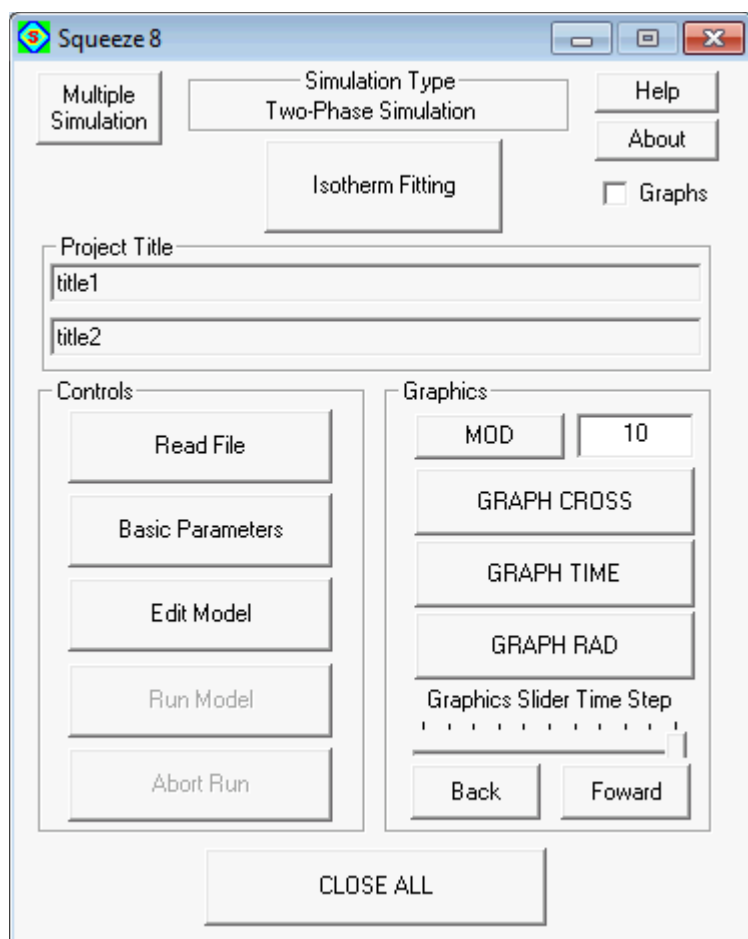


Figure 2.1 The opening page of SQUEEZE V interface

2.1 Multiple Simulations

This button gives the opportunity to simulate automatically any number of projects. This feature is useful for sensitivity studies, where a number of models may have to be simulated. To use this facility the name of the projects should be edited and saved (in Notepad or other text editor), the file should be saved with the extension “inp”. An example is shown in Figure 2.2.

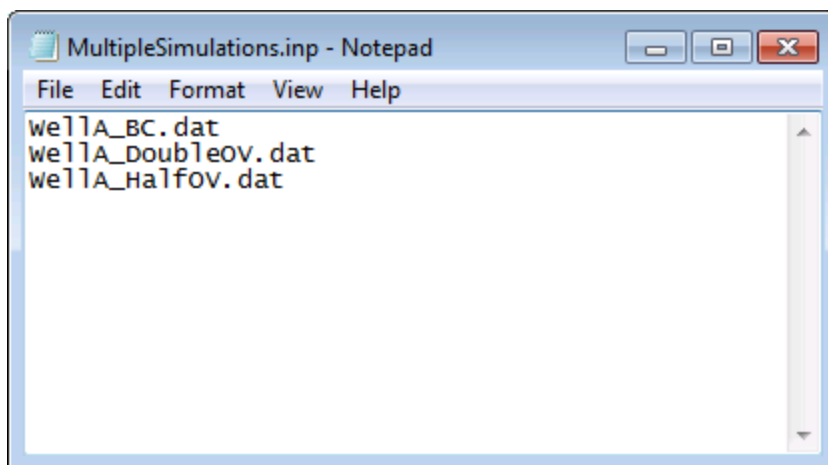


Figure 2.2 Multiple simulations file with “.inp” extension.

2.2 Simulation Type

This text box shows the type of simulation for a given project: Single or Two-phase flow, see Figure 1.1 for model overview.

2.3 Help Button

By pressing the “Help” button the pdf manual appears.

2.4 About Button

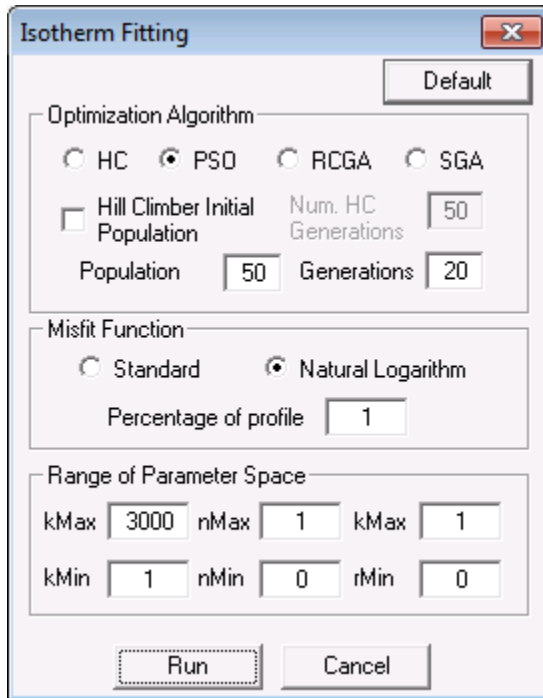
By pressing the “About” button the version of the code will appear. The code is continuously revised and updated. New releases are available to the sponsors through the FAST group website, <http://dscale.org>.

2.5 Isotherm Fitting Button

The ultimate goal of using the SQUEEZE program is to model, predict and design/improve scale inhibitor downhole squeeze treatments. One possible approach, assuming that no experimental data is available but that we have some field inhibitor return profiles, is to “history match” the return profile and then use it for prediction. Previously, the “history match” process was done manually by trial and error until the adsorption parameters that give the best fit to the inhibitor return concentration profile were obtained. This functionality provides an automatic method. When the “Isotherm Fitting” button is pressed a Windows Open File Dialogue appears and a project input file can be loaded. Then the user will be prompted with “Isotherm Fitting” dialogue, Figure 2.3, which shows a number of methods for the automatic isotherm history matching. The user may press the “Default button”, which defines a set of parameters that may be used for most routine calculations.

Once the calculations have finished a window is displayed containing a number of possible isotherm candidates, see Figure 2.4. This is due to the stochastic and iterative nature of the algorithms. The algorithms are population-based; where at each iteration the population is modified, evolving towards the best solutions. The best solution per generation is displayed, giving the user the opportunity to explore a number of possible

solutions. At the end of the process the user must still use their experience and judgement to select the most appropriate isotherm, and may even choose to manually modify the isotherm. However, this automated approach allows the user to quickly identify what will be the eventual isotherm, or at least a close approximation.



The 'Isotherm Fitting' dialog box contains the following settings:

- Optimization Algorithm:**
 - ☐ HC
 - ☒ PSO
 - ☐ RCGA
 - ☐ SGA
- Hill Climber Initial Population:**
 - ☐ Hill Climber Initial Population
 - Num. HC Generations:
 - Population:
 - Generations:
- Misfit Function:**
 - ☐ Standard
 - ☒ Natural Logarithm
 - Percentage of profile:
- Range of Parameter Space:**
 - kMax: nMax: kMax:
 - kMin: nMin: rMin:
- Buttons:** Run, Cancel

Figure 2.3 Isotherm fitting dialogue.

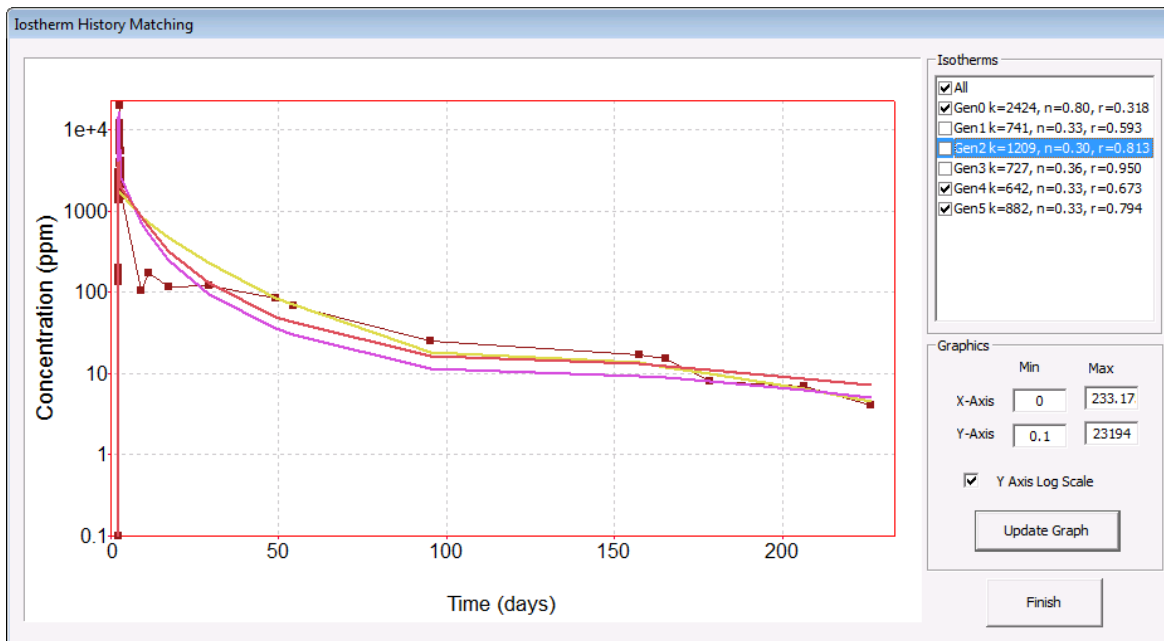


Figure 2.4 Isotherm history matching dialogue.

2.6 Project Title

This text box shows the simulation title, normally used to describe the simulation project.

2.7 Controls Functionality Grouping

This group of buttons contains the functionalities to open, edit and run simulation projects. This will be described below.

2.7.1 Read File Button

This button loads an old project simulation model. Backward compatibility as far as SQUEEZE 5 is maintained. The input data files have a “.dat” extension.

2.7.2 Basic Parameters Button

This functionality determines the basic settings of the simulation project, such as, among others simulation type (single or two-phase), units and title. Depending on the simulation type, the various model types (radial, coreflood, isotherm derivation, etc) are either available or greyed-out, with the appropriate functionalities become active; see Figure 2.5 for an example.

Basic Model Parameters

Simulation Type
☐ Single Phase Simulation ☒ Two Phase Simulation

Project Title
 Project Title1 : title1
 Project Title2 : title2

Units
☐ Metric ☒ Conventional

Model Type
☒ Radial
☐ Linear Core Flood
☐ Isotherm Derivation and Validation
☐ Isotherm Derivation
☐ Analytical Linear

Basic Model Parameters
 Number of Blocks 10
 Number of Layers 4
 Number of Components 1
 Number of Stages 1

Isotherm Derivation Parameters
 VI, Volume of inhibitor injected (cm3) 10
 Vd, Total dead Volume (cm3) 10
 CI, Injected Inhibitor Concentration (ppm) 10
 CShut, Inh. Conc. of the first postflush sample (ppm) 10
 Rho, Density of rock grain (g/cm3) 2.65
 Input Effluent Data Show Data

Well Friction
☐ Well Friction
 Tubing Inner Diameter, ft 0.1
 Tubing Absolute Roughness, ft 0.00015
 Fluid Density, lb/ft3 63

Temperature Calculations Parameters
☐ Temperature Calculations
 Formation Temperature Before Squeeze Treatment, degC 111
 Formation Underburden/Overburden Thermal Conductivity, Btu/ft-day-degC 46
 Thermal Conductivity of Reservoir, Btu/ft-day-degC 45
 Aqueous Heat Capacity, Btu/lb-degC 1.805
 Rock Heat Capacity, Btu/lb-degC 0.42
 Aqueous Phase Density, lb/ft3 62.35
 Rock Grain Density, lb/ft3 165.3

UPDATE CANCEL

Figure 2.5 Basic model parameters dialogue.

- **Simulation Type**

This is the primary setting that determines the model types that will be used in the simulation project. A choice of single phase (water) or two phase (water and oil) may be made.

- **Model type**

The choice of model type depends on the simulation type; refer to Figure 1.1 for model overview. Single phase simulation options are as follows:

- i. Radial, used to simulate field squeeze treatments, modelling the area around the wellbore.
- ii. Linear Coreflood, used to simulate a coreflood experiment.
- iii. Isotherm Derivation and Validation, used to derive an isotherm for a coreflood and validate it against the return profile.

- iv. Isotherm Derivation, used to derive isotherm from a coreflood experiment.
- v. Analytical Linear, used to present an analytical solution to the one-dimensional (1D) inhibitor transport equation which is valid for fractured wells within certain limits.

For Two-phase flow simulation:

- i. Radial, used to simulate field application, modelling the area around the wellbore.
- ii. Linear Coreflood, used to simulate a coreflood experiment.

- Units

Two unit conventions are available, metric and conventional units, which are detailed in Table 2.1.

		Conventional	Conversion Factor →	Metric
Length, height, radius		ft	*0.3048	m
Viscosity		cP	* 1	cP
Volume		bbl	* 0.158987295	m ³
Absolute permeability		mD	* 1	mD
Concentration		ppm	* 1	ppm
Adsorption		mg / l of rock	* 1	mg / l of rock
Stage Type				
Injection/Overflush	Flow rate	bbl/min	* 0.158987295	m ³ /min
Injection/Overflush	Time	hours	* 1	hours
Production	Flow rate	bbl/day	* 0.158987295	m ³ /day
Production	Time	day	* 1	day
Shut-in	Time	hours	* 1	hours

Table 2.1 Metric and conventional units convention.

- Isotherm Derivation Parameters

In this group box the parameters for isotherm derivation are specified:

- iii. Volume of inhibitor injected
- iv. Total dead volume
- v. Injected inhibitor concentration

- vi. Inhibitor concentration of the first postflush sample
- vii. Rock density
- viii. Effluent data

- **Basic Model Parameters**

In this group box, the following basic parameters to run a given simulation are entered:

- i. Number of blocks: Define the number of grid blocks to be used in the simulation and their sizes. Generally more grid blocks to simulate a given volume will produce more accurate results, but will require more time to perform the calculations, see Figure 2.6.

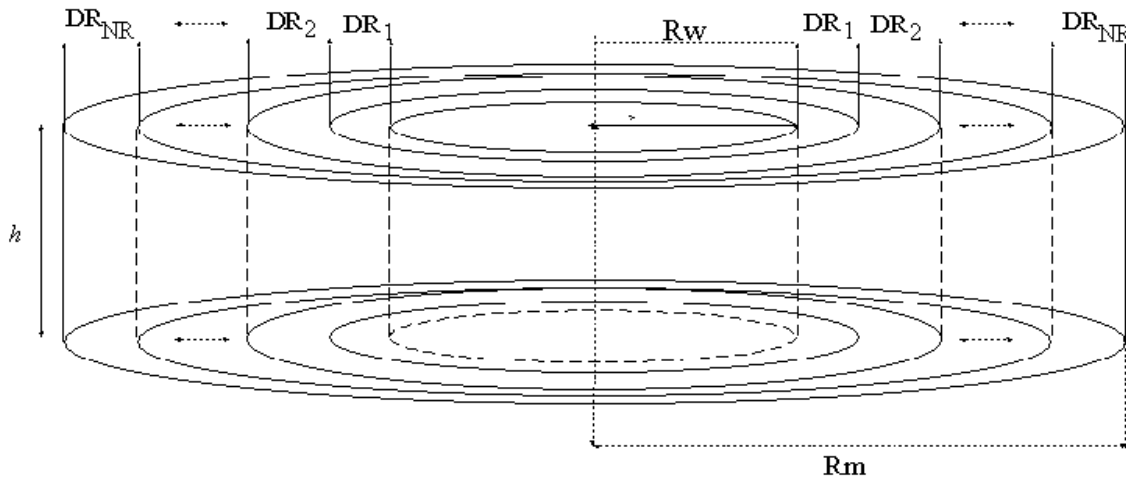


Figure 2.6 Grid and well dimensions - Wellbore radius (R_w) and the maximum radius (R_m).

- ii. Number of layers and thickness (h) of each layer.
- iii. Number of components: In single phase simulation only one component is allowed, whereas in two phase simulation any number is allowed. Thus you could explore the effect of a viscosifying agent on the squeeze lifetime, as an example. Any combination is allowed; for example surfactant plus viscosifier plus bridging agent if desired.
- iv. Number of stages, for example main treatment, overflush, production, etc.

- **Well Friction**

This model is only available for two-phase flow simulations and simulates the frictional pressure drop due to tubing roughness. This is kind of phenomenon occurs generally in long horizontal wells. For preliminary calculations it is advisable to use the defaults values for fluid density and tubing absolute roughness.

- **Temperature Calculation Parameters**

The temperature calculation option is only available for single phase modelling, and must be used if the precipitation option is chosen. The parameters necessary are as follows:

- i. Formation temperature before squeeze treatment

- ii. Formation underburden/overburden thermal conductivity
- iii. Thermal conductivity of reservoir rock
- iv. Aqueous Heat Capacity
- v. Rock heat capacity
- vi. Aqueous phase density
- vii. Rock grain density

2.7.3 Edit Model

When the user presses this button, a new dialogue appears where the layers, components, stages and graphics options are determined; see Figure 2.7.

Edit Model

Layer Properties | Components Properties | Stages Properties | Graphics Options

Formation Properties

Well Radius, ft: 0.5 Maximum Radius of Formation, ft: 70

Layer	Porosity	Absolute Permeability, mD	Height, ft
1	0.23	170	50
2	0.21	200	50

Fluid Properties

☐ All Layers Same Water Viscosity, cP: 0.9 Oil Viscosity, cP: 0.8

Layer	Type	a	Alpha	b	Beta	Irr. Water (Swi)	Irr. Oil (Sor)	Layer Water Sat
1	Corey	0.5	3	0.9	2	0.21	0.18	0.21
2	Corey	0.5	3	0.9	2	0.21	0.18	0.7

OK Cancel Apply Help

Figure 2.7 Edit model dialogue

- Layer properties

The first tab in the dialogue is for the layer properties. Depending on the simulation type, certain information is required; (if it is not required it will be greyed out). The top part of the window is where the formation properties are determined, where we differentiate between radial model (field case) and linear coreflood, which is common for single and two phase simulation.

Formation Properties in Radial Model:

To fully describe the formation properties around the well bore inputs are required for the well radius and the maximum radius of formation, Rad. Max, see Figure 2.8. The latter should be set somewhat beyond the penetration of the injected fluid volume, including preflush, inhibitor and overflush. This can be calculated using Excel based tool, slug-depth.xls located in the Tools directory or downloadable from dsale.org. The number of layers is set in the Basic Parameters dialogue. For multilayer systems porosity, height and permeability are necessary. For single layer systems, permeability is not necessary as it only used to calculate flow distribution between layers.

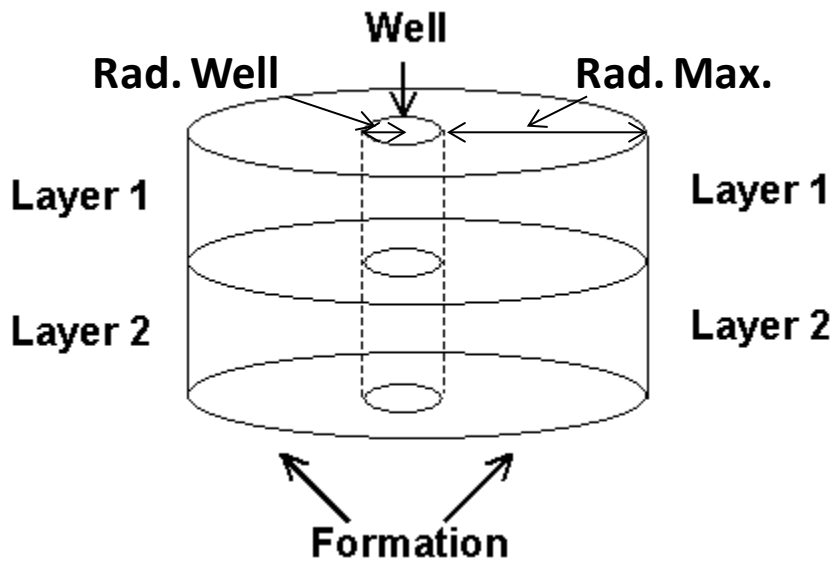


Figure 2.8 Radial model formation properties (field case).

Formation Properties in Linear Core-flood Model:

This model extends the linear 1D equations described by Hong and Shuler (1988) to describe both equilibrium and non-equilibrium inhibitor adsorption squeeze treatments at the laboratory core scale. To fully describe the system length, core diameter and porosity need to be set.

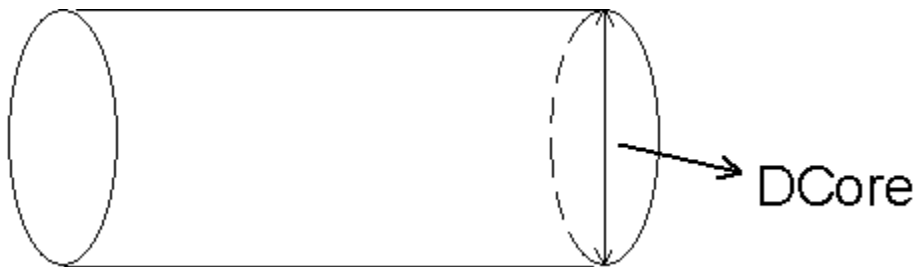


Figure 2.9 Linear coreflood model formation properties (field case).

Fluid Properties

Fluid properties have to be defined when two-phase flow simulation is selected, see Figure 2.10. The simultaneous flow of two immiscible fluids is determined by their viscosities and the relative permeability curves. The fluids are considered to be incompressible, so at any given time and location the addition of both saturations has to be 1 (or 100%).

Edit Model

Layer Properties | Components Properties | Stages Properties | Graphics Options

Formation Properties

Well Radius, ft: 0.5 Maximum Radius of Formation, ft: 70

Layer	Porosity	Absolute Permeability, mD	Height, ft
1	0.23	170	50
2	0.21	200	50

Fluid Properties

☐ All Layers Same Water Viscosity, cP: 0.9 Oil Viscosity, cP: 0.8

Layer	Type	a	Alpha	b	Beta	Irr. Water (Swi)	Irr. Oil (Sor)	Layer Water Sat
1	Corey	0.5	3	0.9	2	0.21	0.18	0.21
2	Corey	0.5	3	0.9	2	0.21	0.18	0.7

OK Cancel Apply Help

Figure 2.10 Fluid properties (two-phase simulation).

The relative permeability curves may be defined by the Corey constants (a, α) for water and (b, β) for oil, see Figure 2.11, where S_{wi} is the irreducible water saturation and S_{ro} is the residual oil saturation, or as a table consisting of three columns, the water saturation, and the water and oil relative permeability values, see Figure 2.12. Any new relative permeability table should be located in the “Relative Permeability” folder using the given format.

$$K_{roil} = b \cdot (1 - S_{or} - S_w)^\beta \quad K_{rwater} = a \cdot (S_w - S_{wi})^\alpha$$

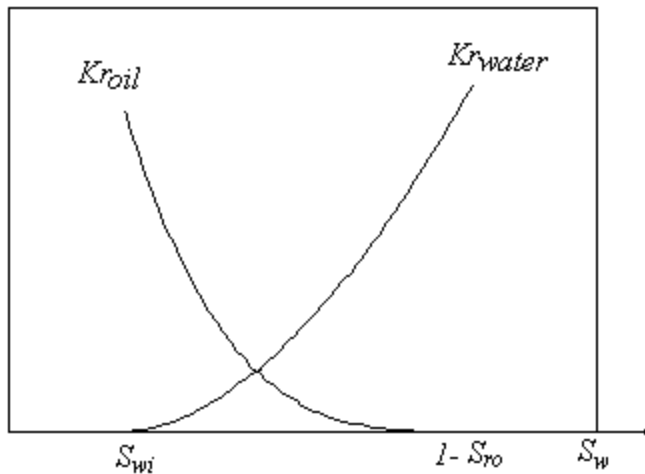


Figure 2.11 Corey relative permeability curves.

RelPerm-Template.txt - Notepad

Num	Krw	Kro
11		
Sw		
0	0	1
0.1	0.1	0.9
0.2	0.2	0.8
0.3	0.3	0.7
0.4	0.4	0.6
0.5	0.5	0.5
0.6	0.6	0.4
0.7	0.7	0.3
0.8	0.8	0.2
0.9	0.9	0.1
1	1	0

Figure 2.12 Relative permeability table.

Finally, to fully describe the system the initial water saturation per layer has to be specified. This can be calculated from the water cut at the time of the treatment. See Figure 2.13 below as an example; imagine the relative permeability curves are defined by the Corey constants or an equivalent table, see Figure 2.13. To calculate the water fractional flow, which is equivalent to water cut, we use the equation below. As you can see, the water fraction flow is function of the water saturation. Figure 2.14 shows graphically the estimated water saturation at 50% water cut, which gives an approximate initial water saturation of 0.56 (or 56%). This file “RelPerm.xls” can be found in the “Relative Permeability” folder or is downloadable from dScale.org.

$$\text{Water Cut} = f_w(S_w) = \frac{1}{1 + \frac{\mu_w \cdot k_{ro}(S_w)}{\mu_o \cdot k_{rw}(S_w)}}$$

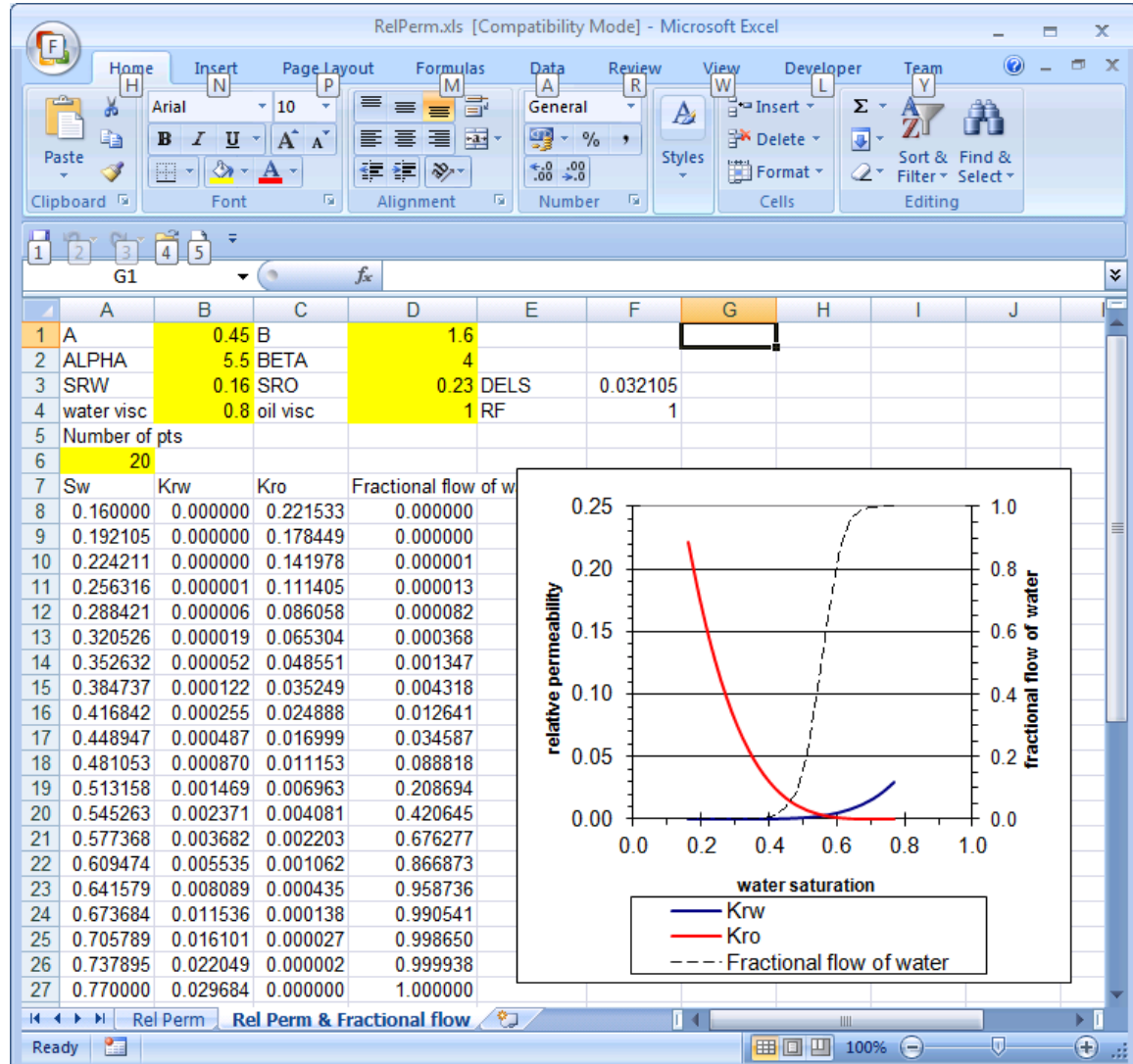


Figure 2.13 Relative permeability curves and fractional flow.

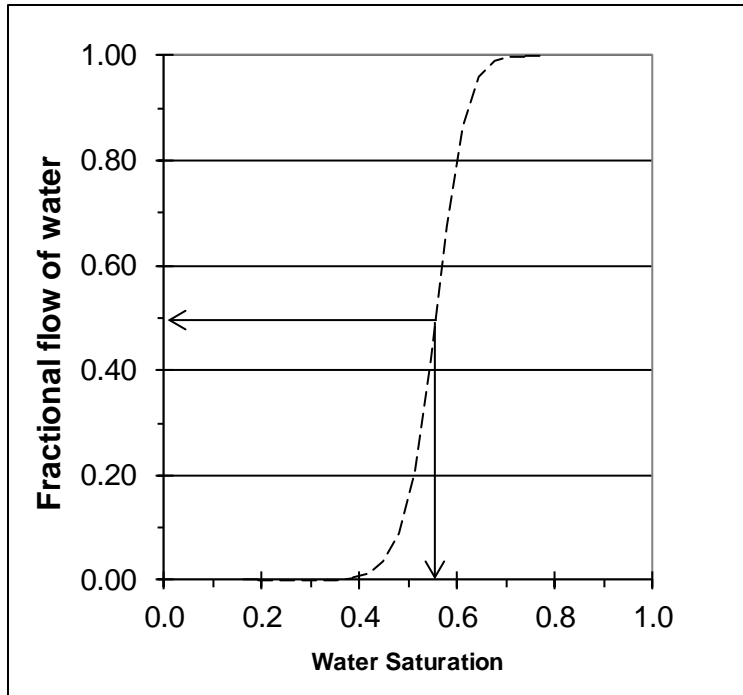


Figure 2.14 Water saturation estimation from water cut. In this example, if water cut is 50%, then water saturation in rock around the well will be 0.56.

- **Component Properties**

This dialogue requires different data for single or two phase simulation, as shown below. Its primary purpose (and sole purpose in single phase simulation) is to define scale inhibitor retention. In two-phase flow simulation, the user can also define the role of a number of other components, in single-phase only scale inhibitor is allowed.

Same all layers:

If this box is ticked all the layers will share the same component properties, this applies for both single and two-phase simulations.

Single Phase Simulation Dialogue:

In Single-Phase simulations, there are two possible models to simulate the scale inhibitor retention, as shown in Figure 2.15, precipitation or adsorption. These can be different for different layers:

- i. **Precipitation:** The dynamic behaviour of a precipitation system is assumed to be governed by the inhibitor solubility and the rate of precipitation/dissolution. The user has to input the critical temperature, Cr.Temp, the inhibitor solubility at temperature equal to or above the Cr.Temp, Cps1, and inhibitor solubility at temperature below Cr.Temp, Cps2.
- ii. **Adsorption:** The dynamic behaviour of an adsorption system is determined by the adsorption isotherm. The user has to input the isotherm form, which can be taken as Freundlich, Langmuir or a table of numbers. The adsorption process may be at equilibrium or non-equilibrium; in the latter case the adsorption rate is required.

Edit Model

Layer Properties | Components Properties | Stages Properties | Graphics Options

Same all layers

☐ All Layers the same

Mass Transfer between Phases

Water Components Adsorption Characteristics

Layer	Prep/Adso	Cr.Temp	Cps1	Cps2	Prep.Type	Rate (1/day)
1	Precipitation	70	80	1e+006	Kinetic1	0.1
Layer	Prep/Adso	Isotherm	k	n	Adso.Type	Rate (1/day)
2	Adsorption	Freundlich	500	0.4	Kinetic1	0.1
Layer	Prep/Adso	Isotherm	Iso Table		Adso.Type	Rate (1/day)
3	Adsorption	IsoTable	TableAdso.txt		Equilibrium	

Oil Components Adsorption Characteristics

OK Cancel Apply Help

Figure 2.15 Single phase component properties

Two-Phase Simulation Dialogue:

There are a number of fundamental differences between single and two-phase simulation in terms of component modelling. In two-phase simulations mass transfer is allowed between phases, as well as there being no restriction to the number of components, which can have different roles, see Figure 2.16 for an example. As before, component properties may vary between different layers.

Edit Model

Layer Properties | **Components Properties** | Stages Properties | Graphics Options

Same all layers

☒ All Layers the same

Mass Transfer between Phases

Comp	Mass Trans	Ratio
1	None	
2	None	

Water Components Adsorption Characteristics

Layer	Comp	Role	Isotherm	k	n	Adso.Type	Rate (1/day)
1	1	Inhib.	Freundlich	500	0.3	Equilibrium	
1	2	Visco.				No Adso	
1	3	Surfact				No Adso	

Oil Components Adsorption Characteristics

Layer	Comp	Role	Isotherm	k	n	Adso.Type	Rate (1/day)
1	1	Inhib.	Freundlich	350	0.3	Equilibrium	
1	2	Visco.				No Adso	
1	3	Surfact				No Adso	

OK Cancel Apply Help

Figure 2.16 Two-phase component properties.

Mass transfer between layers:

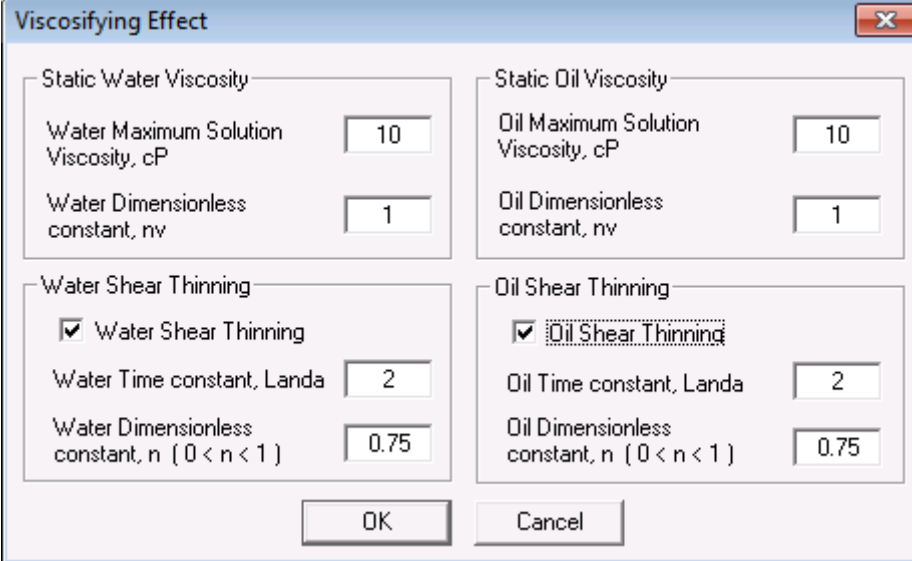
The user defines for a given component whether or not it may be partitioned between the water and the oil phases. The mass transfer between oil and water phases is given by a proportionality constant k_{ow} , which gives the proportion of the concentration in the oil to the water phase, as follows:

$$k_{ow} = \frac{C_o}{C_w}$$

Component Role:

- Tracer: An inert chemical agent, which neither adsorbs onto the rock surface nor is transferred between phases.
- Inhibitor: A chemical agent that may adsorb onto the rock surface and may be transferred between phases.

- iii. Viscosifier: A chemical agent in solution which may increase the viscosity of the solvent (water phase). In addition, it may also adsorb onto the rock surface and may be transferred between phases. The user is required to enter information about the static viscosity and flowing viscosity:



The dialog box titled "Viscosifying Effect" contains four input sections for water and oil properties. Each section has a title, a label, a value input field, and a unit. The "Static Water Viscosity" section has "Water Maximum Solution Viscosity, cP" (10) and "Water Dimensionless constant, nv" (1). The "Static Oil Viscosity" section has "Oil Maximum Solution Viscosity, cP" (10) and "Oil Dimensionless constant, nv" (1). The "Water Shear Thinning" section has a checked checkbox for "Water Shear Thinning", "Water Time constant, Landa" (2), and "Water Dimensionless constant, n (0 < n < 1)" (0.75). The "Oil Shear Thinning" section has a checked checkbox for "Oil Shear Thinning", "Oil Time constant, Landa" (2), and "Oil Dimensionless constant, n (0 < n < 1)" (0.75). At the bottom are "OK" and "Cancel" buttons.

Section	Property	Value	Unit
Static Water Viscosity	Water Maximum Solution Viscosity	10	cP
	Water Dimensionless constant, nv	1	
Static Oil Viscosity	Oil Maximum Solution Viscosity	10	cP
	Oil Dimensionless constant, nv	1	
Water Shear Thinning	Water Shear Thinning	<input checked="" type="checkbox"/>	
	Water Time constant, Landa	2	
	Water Dimensionless constant, n (0 < n < 1)	0.75	
Oil Shear Thinning	Oil Shear Thinning	<input checked="" type="checkbox"/>	
	Oil Time constant, Landa	2	
	Oil Dimensionless constant, n (0 < n < 1)	0.75	

Figure 2.17 Viscosifier model dialogue.

- a. Static Viscosity: At a fixed shear rate, the viscosifier solution viscosity is a function of polymer concentration, as shown schematically in Figure 2.18 and described by the equation accompanying, where μ_v is the polymer solution viscosity, μ_v^{Max} is the maximum viscosifier solution viscosity, and C_v and C_v^{Max} are the actual and the maximum viscosifier concentration in solution, respectively and is a dimensionless constant.

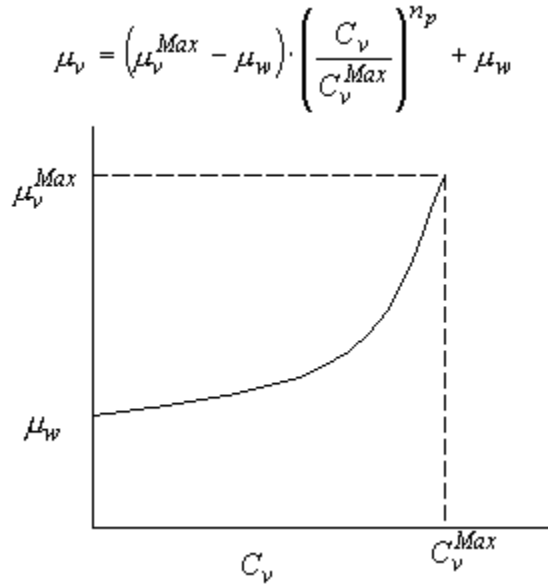


Figure 2.18 Static viscosity model.

- b. Flowing viscosity: The solution viscosity under flowing conditions for a given shear rate may be described by the Carreau model, as given in the equation in Figure 2.19, where $\dot{\gamma}$ is the shear rate, η_o is zero shear rate viscosity, η_∞ is the infinite shear rate viscosity - usually taken as the solvent viscosity, λ is a time constant and n is a dimensionless constant, $0 < n < 1$ ($n = 1$ for Newtonian fluids).

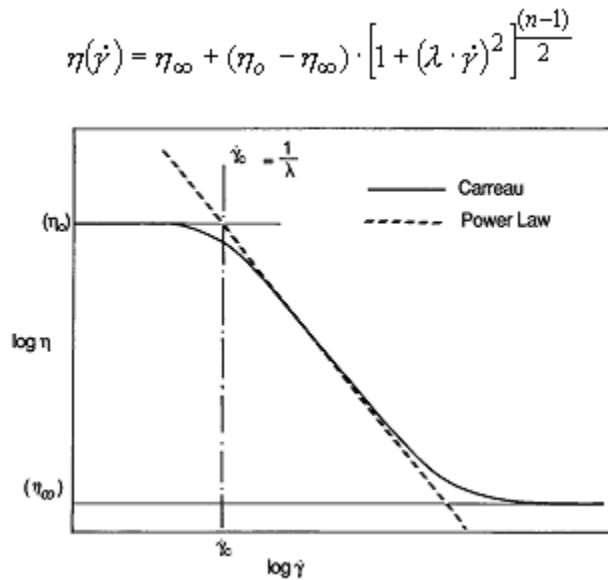


Figure 2.19 Flowing viscosity Carreau Model.

The shear for phase p is calculated with the following equations

$$\dot{\gamma}_p = \left(\frac{3n+1}{4n} \right)^{\frac{n}{n-1}} \cdot \left(\frac{4v_p}{r_{eq}} \right), v_p = \frac{Q_t \cdot f_p}{A \cdot \phi \cdot S_p}, \text{ velocity for phase } p \text{ and } r_{eq} = \sqrt{8K \cdot k_{rp} / \phi \cdot S_p},$$

equivalent radius.

iv. Surfactant: Chemical agent that reduces water and oil residual saturations

a. Reduction of residual water and oil saturations:

The residual water and oil saturations are the saturations at which the phase relative permeabilities become zero, thus making them immobile. This immobility is due to the fact that the water/oil pressure alone is not enough to overcome the relatively higher capillary pressure of the oil/water in the smaller pores.

A surfactant offers the possibility of making these saturations mobile by reducing the oil-water surface tension and thereby reducing the capillary pressure. The effect of the surfactant is modelled through the modification of the relative permeability curves, which are intrinsically linked to the capillary pressures, even if the capillary pressures are not explicitly modelled. Essentially there is a transition between the immiscible (or initial) relative permeability curves related to the original high capillary pressure, and the miscible relative permeability curves related to low capillary pressure. The procedure for applying the surfactant model in SQUEEZE is described below.

Step 1: Calculate the new residual water ($NewS_{wi}$) and oil ($NewS_{ro}$) saturation as a function of the surfactant concentration, C_{Sur} .

$$NewS_{wi} = (S'_{wi} - S_{wi}) \cdot C_{Sur} + S_{wi}, \quad NewS_{ro} = (S'_{ro} - S_{ro}) \cdot C_{Sur} + S_{ro}$$

where $C_{Sur} = \frac{C_w^{Sur} \cdot S_w + C_o^{Sur} \cdot (1 - S_w)}{C_{Sur}^{Max}}$, S'_{wi} and S'_{ro} are the residual

water and oil at maximum surfactant concentrations, C_{Sur}^{Max} and C_w^{Sur} , C_o^{Sur} are the surfactant concentrations in the water and oil phases, since the surfactant may partition between the water and oil phases.

Step 2: Calculate the new value of the water and oil relative permeabilities as functions of the Surfactant concentration, as follows:

New water relative permeability value at a given water saturation:

$$k_{rw} = k_{rw}^{Mis} \cdot W_w + k_{rw}^{Imm} \cdot (1 - W_w)$$

where $k_{rw}^{Mis} = (S_w - NewS_{wi})$ is the miscible water relative permeability curve, k_{rw}^{Imm} is the immiscible (or initial) water relative permeability curve

and W_w is the weight between these two values, with the values calculated as follows:

$$W_w = \begin{cases} 0 & S'_{wi} > S \\ C_{Sur} \cdot \left(\frac{S - S'_{wi}}{1 - S'_{ro} - S'_{wi}} \right)^n & S'_{wi} \leq S \leq 1 - S'_{ro}, \text{ where } n = \frac{1}{40 \cdot C_{Sur} + 10} \\ C_{Sur} & S > 1 - S'_{ro} \end{cases}$$

Figure 2.20 shows the initial (K_{rw}), miscible ($K_{rw} Misc$) and transition ($K_{rw} Trans$) water relative permeability curves.

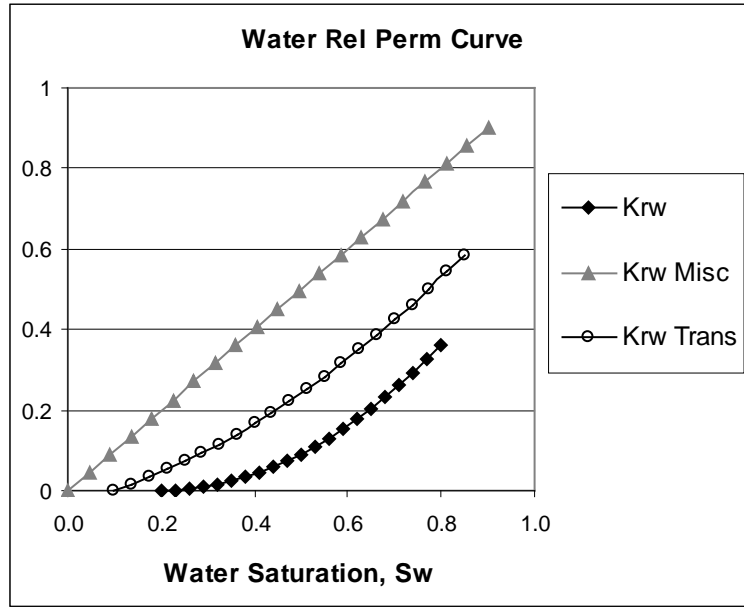


Figure 2.20 Calculation of the water relative permeability.

New Oil Relative Permeability value at a given water saturation:

$$k_{ro} = k_{ro}^{Mis} \cdot W_o + k_{ro}^{Imm} \cdot (1 - W_o)$$

where $k_{ro}^{Mis} = (S_w - New S_{ro})$ is the miscible oil relative permeability curve, k_{ro}^{Imm} is the immiscible (or initial) oil relative permeability curve and W_o is the weight between these two values, values calculated as follows:

$$W_o = \begin{cases} C_{Sur} & S'_{wi} > S \\ C_{Sur} \cdot \left[1 - \left(\frac{S - S'_{wi}}{1 - S'_{ro} - S'_{wi}} \right)^n \right] & S'_{wi} \leq S \leq 1 - S'_{ro}, \text{ where } n = 40 \cdot C_{Sur} + 10 \\ 0 & S > 1 - S'_{ro} \end{cases}$$

Figure 2.21 shows the initial (K_{ro}), miscible ($K_{ro} Misc$) and transition ($K_{ro} Trans$) oil relative permeability curves.

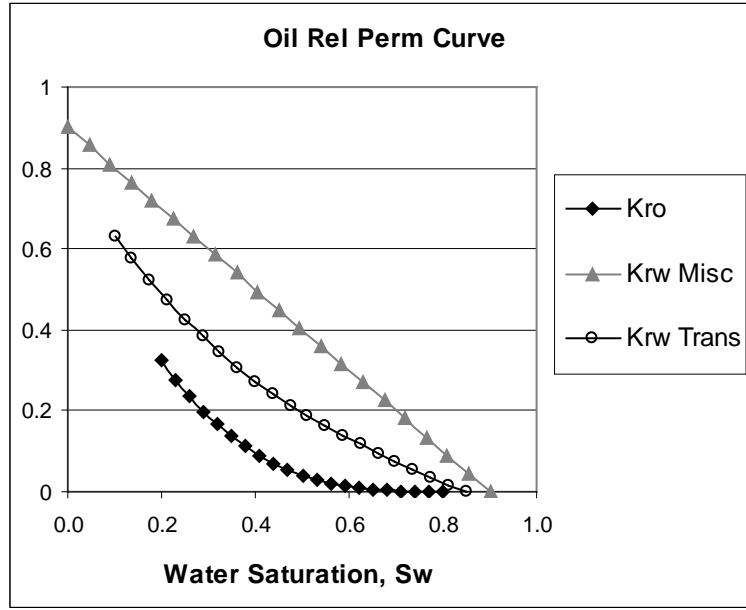


Figure 2.21 Calculation of the oil relative permeability.

b. Alteration of scale inhibitor adsorption:

It is known that inhibitors adsorb less well on hydrophobic surfaces such as those which are present on oil wetted minerals. Previous work has shown that quite often clays are wetted by oil. Since clay minerals in a water wet state are the highest adsorbers, a change of wettability induced by certain surfactants may enhance scale inhibitor adsorption, and so the treatment lifetime. Furthermore, as shown above, the surfactant reduces the residual water and oil saturation, hence augmenting the rock surface available for adsorption. Thus, the SI adsorption alteration is modelled as a function of the augmented rock surface; we use the new residual oil saturation (or augmented water saturation) as the factor that determines it, see Figure 2.22. Assuming high SI adsorption at minimum residual water saturation ($Adso Surf$) and an initial SI adsorption at initial residual water saturation ($Adso No Surf$), where the actual SI adsorption is calculated as an interpolation between both isotherms, as shown below:

$$\Gamma = \left\{ (\Gamma_{Surf} - \Gamma_{Ini}) \cdot \frac{S_w - (1 - S'_{ro})}{(1 - S'_{ro}) - (1 - S_{ro})} + \Gamma_{Ini} \right.$$

where Γ is the actual adsorption, Γ_{Surf} is the new adsorption at maximum surfactant concentration, Γ_{Ini} is the initial adsorption at zero surfactant concentration and S'_{ro} is the residual oil saturation at maximum surfactant concentration, C_{Sur}^{Max} .

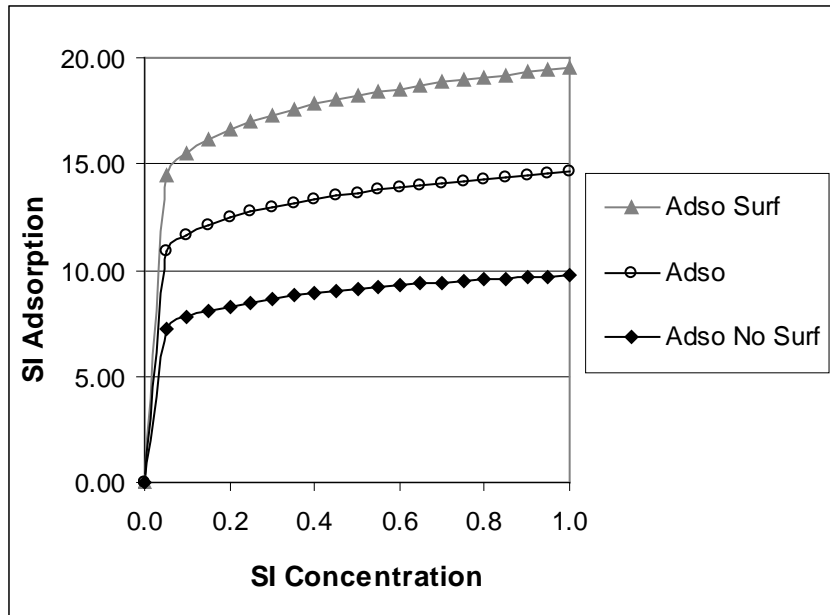


Figure 2.22 SI adsorption isotherm based on the surfactant effect, where *Adso* is the actual adsorption due to the surfactant, *Adso Surf* is the new adsorption at maximum Surfactant concentration and *Adso No Surf* is initial adsorption at zero surfactant concentration.

- v. Blocker: Chemical agent that may reduce the formation permeability due to adsorption, and which may transfer between phases. Although this type of chemical is used for water control treatments, there have been some proposals to use combined SI/blocker treatments.

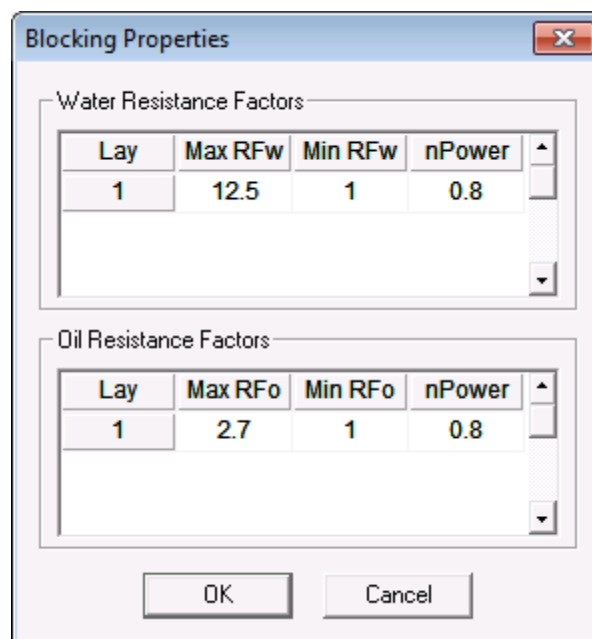


Figure 2.23 Blocker model dialogue.

The permeability reduction is modelled by the application of resistance factors to water and oil flow, which are denoted RF_w and RF_o respectively, see Figure 2.23, with an example of the impact on the relative permeability curves as shown in Figure 2.24.

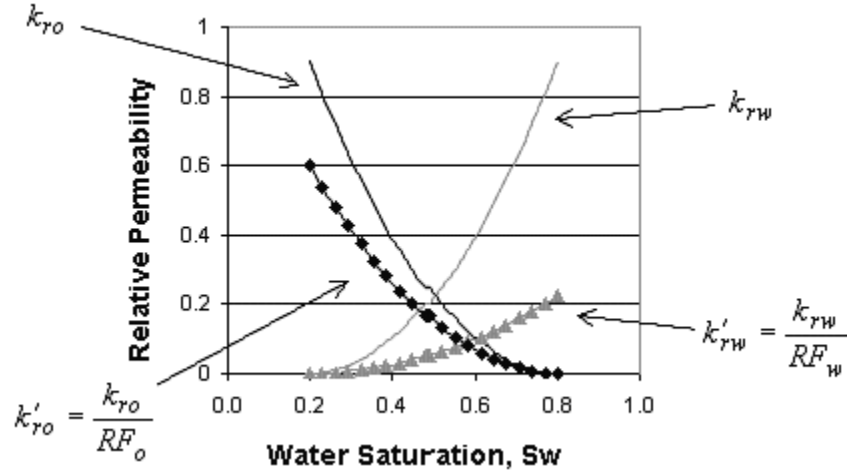


Figure 2.24 Resistant factor model.

To calculate RF_w and RF_o , the variable resistance factor method is used, based on the blocker adsorption level (the higher the blocker adsorption level the greater the layer thickness and hence the greater is the permeability reduction), as shown in the figure below:

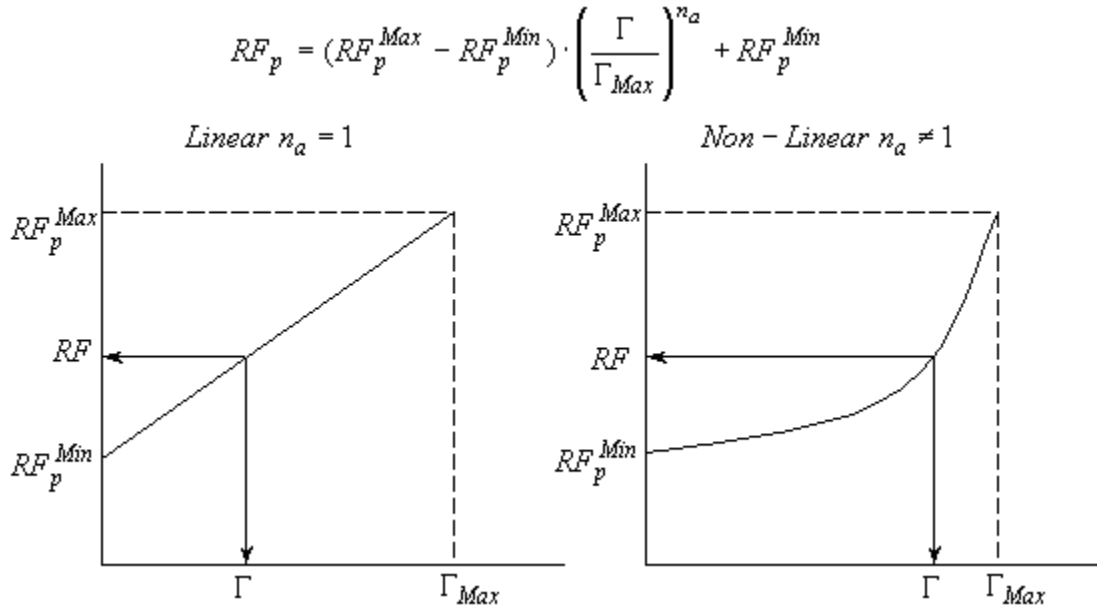


Figure 2.25 Variable resistance factor method

- vi. Bridging agent: This model simulates the enhancement of scale inhibitor retention. Various techniques have been previously proposed such as inducing additional precipitation, injection of surfactant and amphiphilic systems to increase rock surface accessible for adsorption, creation of more positively charged rock surface, incorporating cationic monomer into the structure of the polymeric inhibitor, use of cross-linked scale inhibitor and use of kaolinite for enhanced SI adsorption.

Enhanced Scale Inhibitor Adsorption Models: Based on the published literature about additive packages for SI enhanced adsorption and the different techniques proposed, it is envisaged that the additional SI adsorption will depend on the additive adsorption level. It is also assumed that there is a maximum level of the additive, $\Gamma_{add,max}$, beyond which further SI adsorption enhancement will not occur, which is assumed to occur at the maximum additive adsorption level, and which will provide the maximum enhanced SI adsorption effect. The model to calculate the degree of SI adsorption enhancement is illustrated in Figure 2.26. Figure 2.26 a) and b) show the hypothetical isotherms for SI (Γ_{SI}^* = non-enhanced and Γ_{SI}^{**} = maximum enhanced) and the additive adsorption isotherm, Γ_{add} , respectively. The model calculates the non-enhanced SI adsorption level, $\Gamma_{SI}(\Gamma_{SI})$, the highest possible SI adsorption, $\Gamma_{SI}^{**}(\Gamma_{SI})$, and the adsorption of the additive, $\Gamma_{add}(\Gamma_{add})$. Then by linear interpolation as function of the additive adsorption level, the actual adsorption level of SI, Γ_{SI}^* , is calculated using equation (1), which is graphically represented by Figure 2.26 c).

$$\Gamma_{SI}^*(C_{SI}) = \Gamma_{SI}(C_{SI}) + (\Gamma_{SI}^{**}(C_{SI}) - \Gamma_{SI}(C_{SI})) \cdot \frac{\Gamma_{add}(C_{add})}{\Gamma_{add,max}}$$

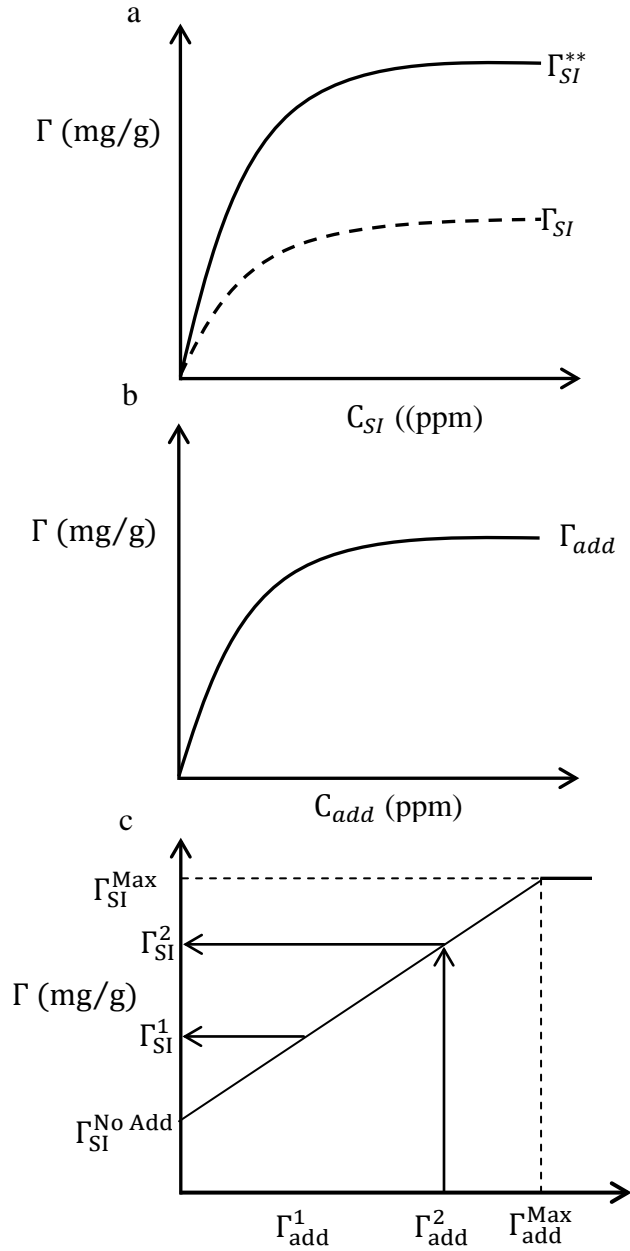


Figure 2.26 Scheme to calculate a degree of SI adsorption enhancement

- *Stages Properties*

This dialogue requires different data for single or two phase simulation as shown below, which is where every stage is defined such as the type of stage (injection or production), component concentration, flow rate, volume or time and time step size. In addition, the injection and production controls are determined, including flow rate distribution, where any layer may be opened or closed at any stage. There are some differences between single and two-phase simulations. For two phase flow, component concentrations in each

phase may be specified, and the fractional flow of water (akin to water cut) must also be entered, as described below.

Single Phase Simulation Dialogue

Figure 2.16 shows the input data dialogue, where the properties of each stage can be determined.

Edit Model

Layer Properties | Components Properties | **Stages Properties** | Graphics Options

Stages

Headers ☒

Enter position of Stage:

Automatic Time Step ☐
☒ Low ☐ High

Stage	Type	TI	Inj/Prod Controls	F/T/V	Flow bbl/min	Time hours	Vol bbl	T. Step hours
1	Inj/Preflush		[SI] 150000	F/V	6	5	1800	0.01
2	Overflush		Click here	F/V	6	5.55556	2000	0.01
3	Shut-in		Pres.Lay			12		0.01
4	Production		Click here	F/V	6000	90	540000	0.1

Totals

Water Volume Injected 3800 bbls in 10.56 hours, Scale Inhibitor Volume Injected 270.00 bbls
 Water Volume Produced 2.97e6 bbls in 360.00 days

Figure 2.27 Single-phase stages properties.

Add/Remove stage:

The user can add or remove a stage at any position. This facility is especially useful if a new stage is to be included in an update of an old design.

Headers

The user can remove the stages headers, so that more stages are visible on the panel.

Stage type:

There are two principal types of stage: injection and production. Within an injection stage, it is possible to differentiate between injection/preflush, and overflush.

- i. Injection/Preflush: Stages where the chemical is injected at low concentration as a preflush or at working concentrations as a main slug.
- ii. Overflush: This assumes the overflush contains zero chemical concentration. If the overflush does contain some chemical then an injection/preflush stage may be selected instead.
- iii. Shut-in: Any period of time during which the well is neither producing nor injecting. This stage allows chemical to adsorb.
- iv. Production: Any period that the well is back in production.

Initial Formation Temperature, TI:

This data is required for temperature calculations (which are necessary for precipitation calculations).

Injection/Production Controls, (Inj/Prod Controls):

This dialogue enables the user to determine a number of properties such as the flow rate distribution and the component concentration in injection/preflush stages.

Injected Concentration and Flow Distribution

Layers

Flow rate in terms of

☒ KH Product

☐ PIDF

☐ Pressurised Layers

☒ All Layers Open

Units Pres Layers

☐ bar ☒ psi

☐ All Press Same

Lay	Open Lay	KH (Auto)	ProdFw
1	Open		
2	Open		
3	Open		

Components Concentrations

☐ All Components the same

Comp	Wat Conc	Oil Conc
(1) Inhib.	150000	0

OK Cancel

Figure 2.28 Single-phase injection/production controls dialogue.

i. All Layers Open:

By setting this control, all the layers are set to be open.

ii. Flow Rate in terms of:

If the flow distribution is known, then this can input by selecting *PIDf* (*PIDf* stands for Productivity Index Dimensionless factor). If it is not know then it will automatically be calculated assuming the fluid is distributed according to the kH fraction, as shown in the equation below, where H is the layer height and k_{eff} refers to the effective permeability of the layer:

$$Q_w(i) = \frac{H(i) \cdot k_{eff}(i)}{\sum_{\text{all-active-layers}} (H \cdot k_{eff})} \cdot Q_T$$

iii. Initial Components Concentrations:

The user has only to input the component concentration in the water phase.

Flow/Time/Volumes, (F/T/V):

The user needs to input two of the following flow rate, time or volume, the third is calculated automatically.

Time Step, (T.Step):

In single phase calculations, the user needs to input a value for the size of each time step per stage. As a guideline, recommended values are:

preflush/injection/overflush	$\Delta t = 0.001$ days
shut-in (equilibrium adsorption)	$\Delta t = 0.10$ days
shut-in (non-equilibrium adsorption)	$\Delta t = 0.02$ days
production (0-2 days)	$\Delta t = 0.01$ days
production (3-10 days)	$\Delta t = 0.05$ days
production (11days-1st month)	$\Delta t = 0.10$ days
production (till end of simulation)	$\Delta t = 0.50$ days

The larger the time-step, the faster the simulation will run. If it is made too large, a convergence error will be reported, and the simulation will abort.

Get Totals:

Get the total injected volume of water, scale inhibitor and produced water.

Two-Phase Simulation Dialogue

Figure 2.17 shows the input data dialogue, where the properties of each stage can be determined.

Edit Model

Layer Properties | Components Properties | **Stages Properties** | Graphics Options

Stages

Headers ☒

Enter position of Stage

Automatic Time Step ☒
☒ Low ☐ High

Stage	Type	Qw/Qt	Inj/Prod Controls	F/T/V	Flow bbl/min	Time hours	Vol bbl	T. Step hours
1	Inj/Preflush	1	[SI] 100000	FV	15	2	1800	
2	Overflush	1	Click here	FV	15	7	6300	
3	Shut-in		Pres.Lay			5		
4	Production		Click here	FV	2400	30	72000	

Totals

Water Volume Injected 8100 bbls in 9.00 hours, Scale Inhibitor Volume Injected 180.00 bbls
 Fluid Volume Produced 1.06e6 bbls in 440.00 days

Figure 2.29 Two-phase stages properties.

Add/Remove stage:

The user can add or remove a stage at any position. This facility is especially useful if a new stage is to be included in an update of an old design.

Headers

The user can remove the stages headers, so that more stages are visible on the panel.

Stage type:

There are two principal types of stage: injection and production. Within an injection stage, it is possible to differentiate between injection/preflush, and overflush.

- i. Injection/Preflush: Stages where the chemical is injected at low concentration as a preflush or at working concentrations as a main slug.
- ii. Overflush: This assumes the overflush contains zero chemical concentration. If the overflush does contain some chemical then an injection/preflush stage may be selected instead.

- iii. Shut-in: Any period of time during which the well is neither producing nor injecting. This stage allows chemical to adsorb.
Production: Any period that the well is back in production.

Injection water flow rate fraction, (Q_{wt}/Q_t):

In two-phase flow simulations the fraction of the total fluid that is water must be set. For instance, if an emulsion is injected with 20% water content, then Q_{wt}/Q_t should be set to 0.2, (1 implies 100% water and 0 implies 100% oil).

Injection/Production Controls, (Inj/Prod Controls):

This dialogue enables the user to determine a number of properties such as the flow rate distribution and the component concentration in the Injection/Preflush stages. Note that for any components to have the desired effect, be it a viscosifier, blocker, etc, the injection concentration has to be set to a finite value.

Injected Concentration and Flow Distribution

Layers

Flow rate in terms of

☒ KH Product

☐ PIDF

☐ Pressurised Layers

☒ All Layers Open

Units Pres Layers

☐ bar ☒ psi

☐ All Press Same

Lay	Open Lay	KH (Auto)	ProdFw
1	Open		
2	Open		

Components Concentrations

☐ All Components the same

Comp	Wat Conc	Oil Conc
(1) Inhib.	100000	0
(2) Visco.	0	0
(3) Bridging	0	0

OK Cancel

Figure 2.30 Single-phase injection/production controls dialogue.

- i. All Layers Open:
By setting this control, all the layers are set to be open.
- ii. Flow rate can be set using one of the following three options:

- a. If the flow distribution is known, then this can input by selecting *PIDf*.
- b. If the flow distribution is known, then this can input by selecting *PIDf* (*PIDf* stands for Productivity Index Dimensionless factor). If it is not know then it will automatically be calculated assuming the fluid is distributed according to the *kH* fraction, as shown in the equation below, where *H* is the layer height and *keff* refers to the effective permeability of the layer:

$$Q_w(i) = \frac{H(i) \cdot k_{eff}(i)}{\sum_{\text{all-active-layers}} (H \cdot k_{eff})} \cdot Q_T$$

- c. Pressurised layers: If the layer pressures are known, then they can be included to calculate the flow distribution. In the current version, crossflow in the well bore is not simulated.

In addition to setting the flow rate, the production fractional flow must also be set *ProdFw*:

It can be thought as the water cut at maximum radius of formation, which normally is assumed to be the same than the well water cut, since it seems reasonable that the amount of water flowing to the well will be very similar few feet in the formation. It only applies to production stages. Its value ranges between 0 and 1 and it can be different for each layer. Figure 2.31 and the equation below illustrate the role of on determining the flow of water in each layer, in terms of the total flow rate, Q_{wi} , $i = 1, 2$.

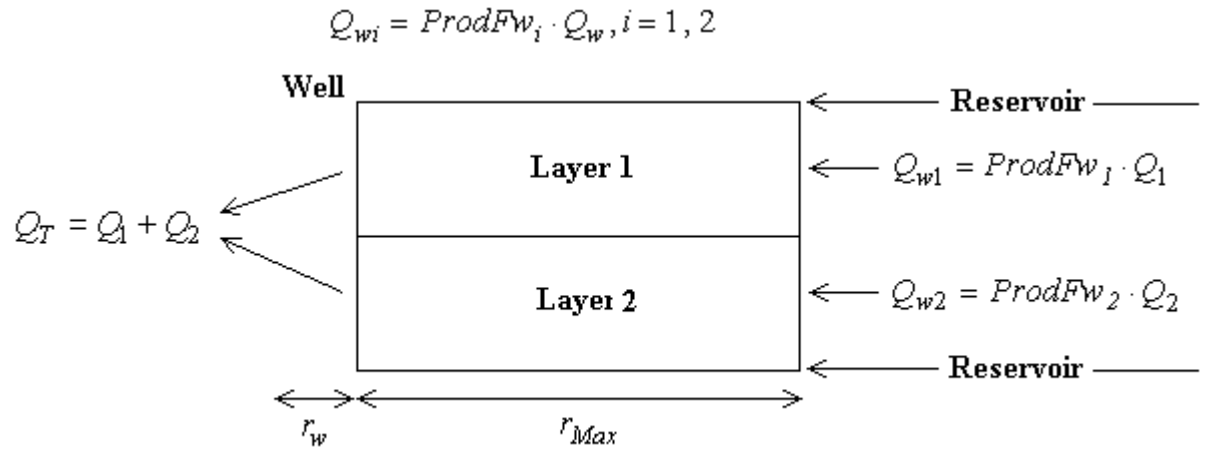


Figure 2.31 Two-phase flow boundary conditions during production.

iv. Initial Components Concentrations:

The user has only to input the component concentration in the water or oil phase, but may do so in both.

Flow/Time/Volumes, F/T/V:

The user needs to input two of the following flow rate, time or volume, the third is calculated automatically.

Time Step, (T.Step):

The time step can be automatically calculated to guarantee numerical stability.

Get Totals:

Calculate and display the total injected volume of water, scale inhibitor and produced water.

- **Graphics Options**

The graphic options are slightly different for single, see Figure 2.32 and two-phase flow simulations, see Figure 2.33. The most important features are the following:

- i. Minimum inhibitor concentration for colour shading, see Figure 2.35 right hand side graph, when completion that are below MIC are coloured red.
- ii. Report up to five squeeze lifetimes at different MICs in output file.
- iii. Include field data in return graph.
- iv. Plot up to 8 previous run, extremely useful for analysis in sensitivity calculations.

Single Phase Flow Simulation

Edit Model

Layer Properties | Components Properties | Stages Properties | **Graphics Options**

Return Curve Plot

Return Concentration/Temperature plotted vs. ☒ Time (linear scale) ☐ Water Volume (linear scale)
☐ Time (log scale) ☐ Water Volume (log scale)

Screen Display showing Conc. and Temp vs. Distance ☐

Maximum Number of Iterations per Time Step Calculation (recommended = 15) Number of Time Steps per report (in the range of 10 to 200)

Inhibitor Threshold Conc. for Colour-Shading. Plot in the Overall Return Curve Window

In Situ Concentration vs. Distance ☐

Report up to five threshold inhibitor concentrations (ppm) at which squeeze lifetime is reported

<input type="text" value="10"/>	<input type="text" value="4.5"/>	<input type="text"/>	<input type="text"/>	<input type="text"/>
---------------------------------	----------------------------------	----------------------	----------------------	----------------------

Solution Convergence Tolerance (Default, Toltr = 1e-8) Recommended Range, Toltr = 1e-6 to 1.0e-10

Field Squeeze Return Data

Graphics Param. for multilayer Simulation

You may plot up to 8 previous simulations on the same graph as the current simulation

CASE3M-2.out	Click here 5
CASE3M-3.out	Click here 6
Click here 3	Click here 7
Click here 4	Click here 8

Enter 0 to remove all.

Figure 2.32 Single-phase flow graphics options dialogue

Two Phase Flow Simulation

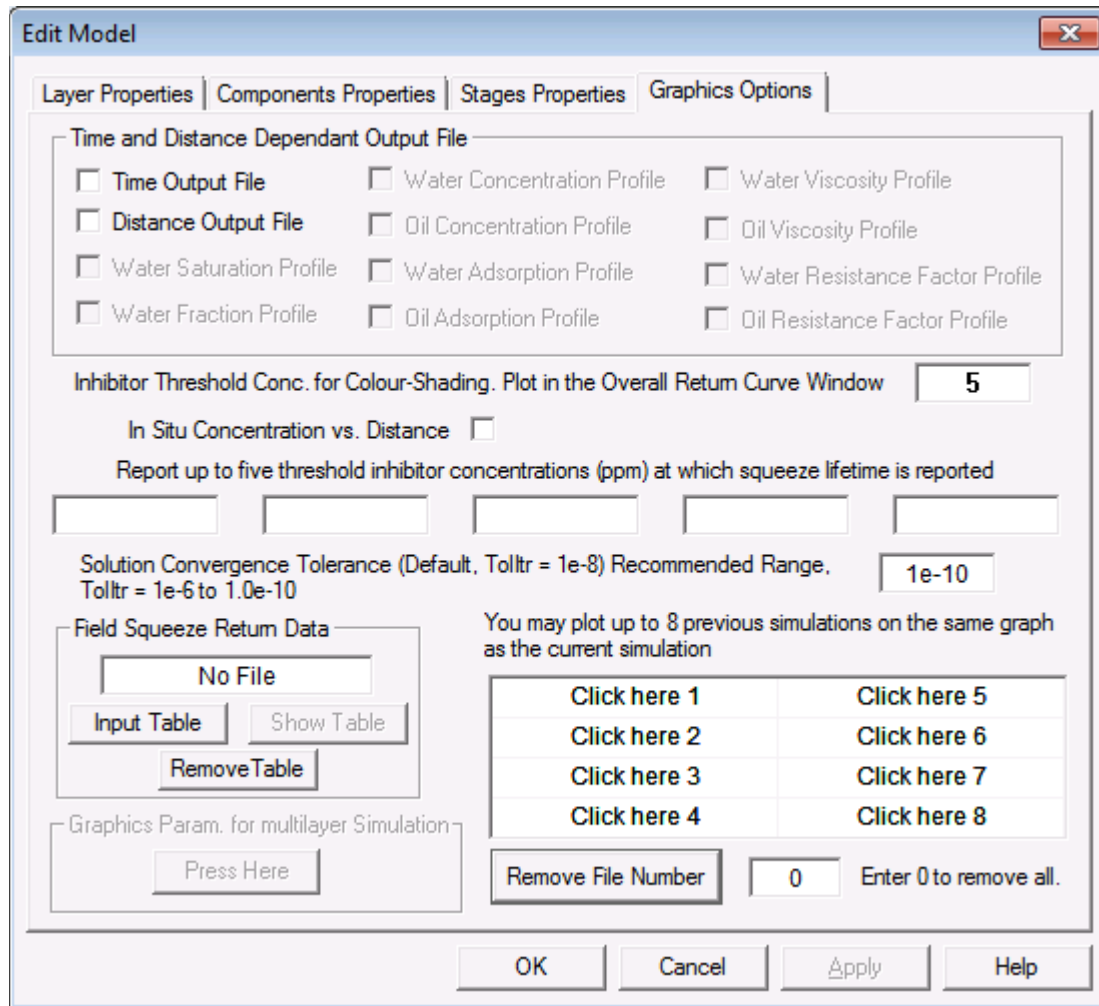


Figure 2.33 Two-Phase Flow Graphics Options Dialogue.

Once data entry in the “Edit Model” panel is completed, pressing the “OK” button will save the changes to file, and return the user to the opening overall control panel. Pressing “Cancel” will return the user to the program control panel without saving any changes. Once back in the main control panel, it is possible to run the program and then subsequently to visualize results.

2.8 Run Model

This option starts the simulation and creates an output file with extension ".out".

2.9 Abort Model

This option aborts the simulation at any time; information generated prior to this is saved to the ".out" file.

2.10 Graphics functionality group

This group box enables the user to navigate through a number of graphs, and is designed to be a postprocessor. Once the simulation is run the user can use the “Graphics Slider

Time Step” to navigate through the reported time steps. The functionalities will be described below.

2.10.1 Output Frequency, (MOD)

Only for two-phase flow simulation, this option sets the number of time steps per report.

2.10.2 GRAPH CROSS

In situ inhibitor concentration field in the near-well formation is displayed along the radial distance from the wellbore. Seven colours are used to represent different concentration intervals. The grid is proportional to the grid block sizes used in the numerical simulation. This graph is very useful to analyse placement or which layers have been depleted of scale inhibitor, see Figure 2.34.

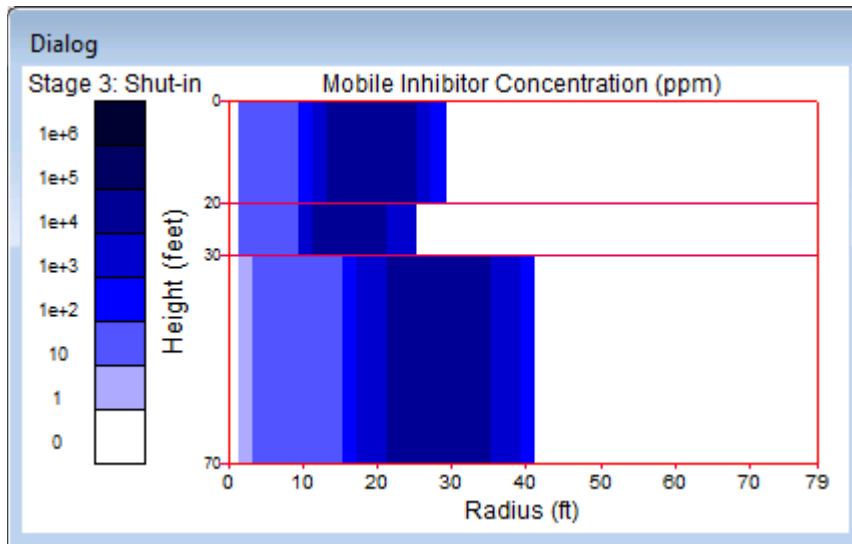


Figure 2.34 The window for the in situ inhibitor properties for the multi-layer radial model.

2.10.3 GRAPH TIME

There are two graphs that are activated with this control. Figure 2.35 shows the scale inhibitor overall return concentration. If there are any measured field inhibitor squeeze return data, these data points also can be (optionally) displayed on the screen. If you have outputs from your previous simulation runs for the same treatment (e.g. with different adsorption isotherm parameters), the return curves for the previous runs can also be displayed along with the currently modeled inhibitor returns. The colour bar appearing on the rightmost side of Figure 2.35, shows the wellbore 'status', i.e., whether the local wellbore block concentration is below, equal to or above a user-specified MIC (GrafMic) as follows:

- i. Red colour signals a wellbore block concentration that is below the MIC (GrafMic);
- ii. Green colour is for concentration greater than the MIC + 1 ppm; and
- iii. Yellow colour is for concentration between the MIC and MIC + 1 ppm.

Thus, the red colour is used to identify possible scaling problems in that perforation, and green colour indicates adequate protection.

The overall inhibitor return curve can be depicted in two ways: the well top inhibitor concentration, C , vs. the treatment time, t , or C vs. the cumulative water volume (including preflush, injection, overflush and produced water volume). The legends (i.e., line style and colour) are chosen to clearly differentiate the various pieces of information which are displayed on the screen. These legends apply to all types of simulations which are described below:

- i. Modelled inhibitor return curve, temperature - *blue solid line*,
- ii. Modelled inhibitor return curve(s) from the previous run(s) - *green, pink, or other coloured solid line*,
- iii. Measured field squeeze return data or core flood effluent data - *red squares*.

Figure 2.36 displays an example layer return graphics on completion of the simulation. It shows the simulated inhibitor returns for three layers. The horizontal axis of the figure represents either the time or the total water production volume which is the same as that in the overall return curve.

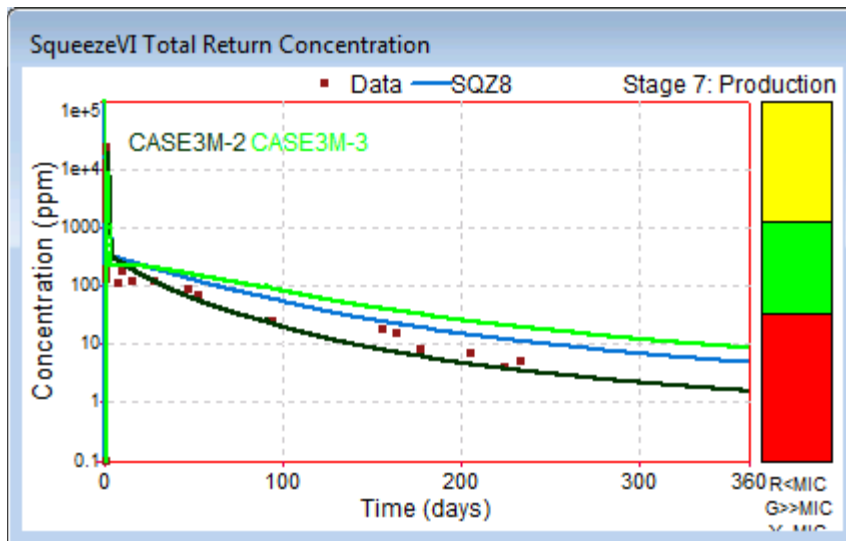


Figure 2.35 The overall return curve window for the multi-layer radial model.

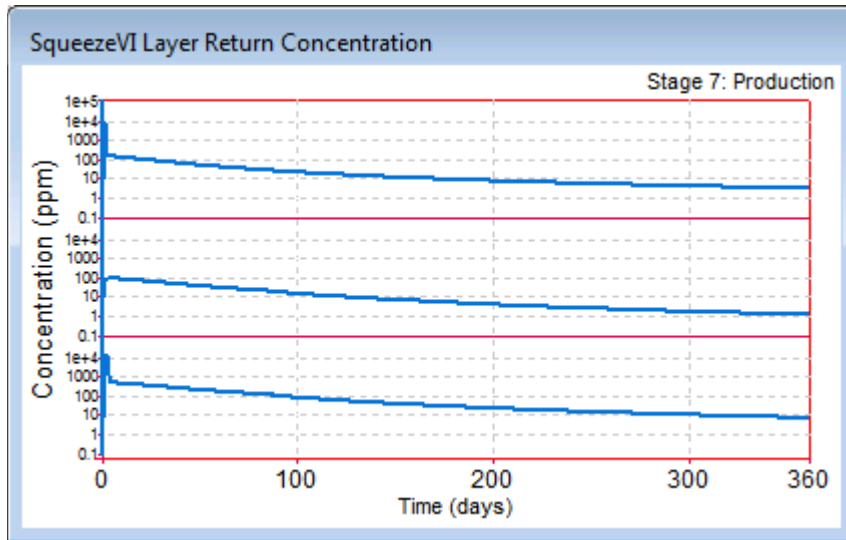


Figure 2.36 The window for the layer inhibitor returns for the multi-layer radial model.

2.10.4 GRAPH RAD

Adsorption and concentration in solution normalised against the maximum adsorption and concentration respectively, see Figure 2.37; and temperature (single-phase) and water saturation (two-phase) along the radial distance for chosen layers, see Figure 2.38.

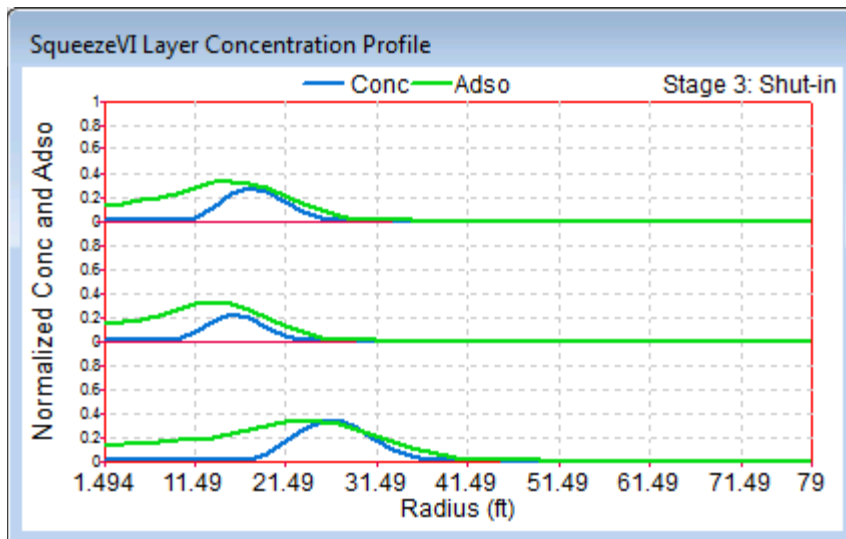


Figure 2.37 Adsorbed and concentration scale inhibitor for the multi-layer radial model.

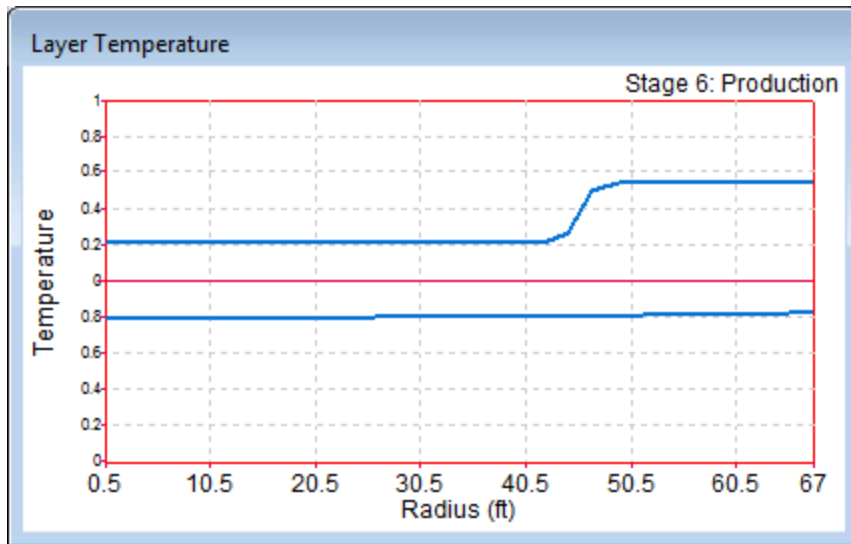


Figure 2.38 Water saturation for the multi-layer radial model.

2.11 CLOSE ALL

This functionality closes all the graphs and the main dialogue.