

## DECKBLATT

Projekt	PSP-Element	Obj. Kenn.	Aufgabe	UA	Lfd. Nr.	Rev.
EU 346	9K	352126.44	EG	RB	0011	00

Titel der Unterlage:

Overview Description of Eclipse 100 Modell

Seite:

I.

Stand:

Feb. 1990

Ersteller:

Intera Inc., Austin, Texas

Textnummer:

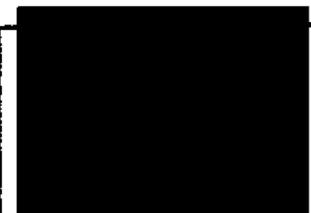
Stempelfeld:

PSP-Element TP...9K/21285

zu Plan-Kapitel: 3.9



Freigabe für Behörden



Freigabe im Projekt

Diese Unterlage unterliegt samt Inhalt dem Schutz des Urheberrechts sowie der Pflicht zur vertraulichen Behandlung auch bei Beförderung und Vernichtung und darf vom Empfänger nur auftragsbezogen genutzt, vervielfältigt und Dritten zugänglich gemacht werden. Eine andere Verwendung und Weitergabe bedarf der ausdrücklichen Zustimmung.



OVERVIEW DESCRIPTION  
OF  
ECLIPSE 100  
MODEL

Prepared by

INTERA Inc.  
6850 Austin Center Boulevard  
Suite 300  
Austin, Texas 78731

February 1990

H00090R035

## 1.0 INTRODUCTION

This document provides a description of the ECLIPSE 100 code in support of the two-phase flow modeling work done for gas release from Konrad Mine. The ECLIPSE model solves three-phase flow (oil, water and gas) equations.

The governing equations, finite-difference approximations, and the major assumptions are presented in Section 2.

A summary of the model features is given in Section 3. Solution method is described in Section 4. The dual porosity model is described in Section 5. Section 6 describes saturation functions and handling of capillary equilibrium. Constituent tracking included in ECLIPSE is discussed in Section 7. Finally, unit conventions are listed in Section 8.

Most of the material is directly photocopied from the ECLIPSE reference manual and the associated appendices. Therefore in places, it may read part of a different document.

## 2.0 MODEL EQUATIONS

ECLIPSE 100 solves a coupled set of equations describing three-phase flow through porous media or fractured rock. Since only the two-phase capability was used in this work, the discussions in this section are limited to modeling of two phases only. The two phases may be a gas and a liquid phase (e.g., water) or two immiscible liquids (e.g., water and an organic solvent). For porous media, both the phases may be present anywhere in the system. In the case of fractured media, the flow system is assumed to consist of a primary flow system, and a secondary flow system. The fractured rock system may be modeled as a dual-porosity or a dual-permeability system. In a dual-porosity, each secondary porosity block communicates with a primary porosity block. Each primary porosity block communicates with all the other primary porosity blocks. But the secondary porosity blocks do not communicate with each other. In a dual-permeability concept, the secondary porosity blocks communicate with each other as well.

The governing equations are described in Section 2.1. The finite difference method used to solve these equations is presented in Section 2.2. The governing flow equations for the porous media and numerical approximations are widely accepted and used in petroleum reservoir simulation. Section 2.3 presents the assumptions and specifications for the solution scheme used in the code.

### 2.1 GOVERNING EQUATIONS

The governing equations to be solved in this code are derived from conservation principles of mass and momentum. The net flux of mass or momentum in or out of a control volume must equal the time rate of change of that quantity. Simply stated:

$$\text{FLUX IN} - \text{FLUX OUT} = \text{ACCUMULATION}$$

In the following subsections, the flow equations for the two phases are derived.

The conservation of mass equations for the water and gas phases for flow through fractured or porous media can be written as:

$$-\nabla \cdot (\rho_w \underline{u}_w) - q_w = \frac{\partial}{\partial t} (\phi \rho_w s_w) \quad (\text{Water}) \quad (1)$$

The gas component can exist both in the gas phase and in dissolved form in the water phase.

$$-\nabla \cdot [R_s \rho_w \underline{u}_w + \rho_g \underline{u}_g] - q_g - R_s q_w = \frac{\partial}{\partial t} [\phi (R_s \rho_w s_w + \rho_g s_g)] \quad (2)$$

where

$\rho_w, \rho_g$  = densities of water and gas

$\underline{u}_w, \underline{u}_g$  = Darcy (superficial) velocities of water and gas

$s_w, s_g$  = saturations of water and gas

$q_w, q_g$  = mass source/sink for water and gas

$R_s$  = amount of gas dissolved in water (gas-water ratio in dissolved phase)

$\phi$  = porosity

$t$  = time.

The underline represents a vector quantity. In the water equation; the first term is the convection term, the second term represents sources and sinks; the right hand side of the equation represents the accumulation term where porosity, density and saturation may change with time. In the gas equation, the first term (within the square bracket) is the convection term. The first part is convection of dissolved gas in water, and the second part is convection of free gas. The second term in the gas equation

represents sources and sinks of free gas. The third term represents sources and sinks of gas dissolved in water. The right hand side is the accumulation term. The first part is the accumulation of dissolved gas and the second part is the accumulation of free gas. The solution gas ratio  $R_g$  is a function of water-phase pressure. Free gas can exist only if water is fully saturated with dissolved gas at the given pressure.

In porous media, inertial effects can usually be ignored so that the momentum equation for two-phase flow are given by extension of Darcy's law:

$$\underline{u}_w = \frac{- \underline{k} k_{rw}}{\mu_w} \nabla(p_w - \rho_w g h) \quad (\text{Water}) \quad (3)$$

and

$$\underline{u}_g = \frac{- \underline{k} k_{rg}}{\mu_g} \nabla(p_g - \rho_g g h) \quad (\text{Gas}) \quad (4)$$

where

$\underline{k}$  = bulk intrinsic permeability tensor for an anisotropic medium,

$p_w, p_g$  = pressures of water and gas phases,

$k_{rw}, k_{rg}$  = relative permeabilities for water and gas; functions of saturation,

$\mu_w, \mu_g$  = dynamic viscosities of water and gas,

$h$  = depth below datum elevation

$g$  = gravitational constant.

Equations (3) and (4) are substituted into Equations (1) and (2), respectively, to obtain:

$$\nabla \cdot \left[ \frac{k}{\mu_w} \frac{k_{rw} \rho_w}{\mu_w} \nabla (p_w - \rho_w g h) \right] - q_w = \frac{\partial}{\partial t} (\phi \rho_w s_w) \quad (5)$$

$$\nabla \cdot \left[ R_s k \frac{k_{rw} \rho_w}{\mu_w} \nabla (p_w - \rho_w g h) + k \frac{k_{rg} \rho_g}{\mu_g} \nabla (p_g - \rho_g g h) \right] - q_g - R_s q_w = \frac{\partial}{\partial t} (\phi R_s \rho_w s_w) + \frac{\partial}{\partial t} (\phi \rho_g s_g) \quad (6)$$

The equations (5) and (6) are solved simultaneously using the relationships described in the following subsection.

### 2.1.1 Coupling of Flow Equations

Equations (5) and (6) are the partial differential equations for describing flow of water and gas. The unknowns in these two equations are  $p_w$ ,  $p_g$ ,  $s_w$ ,  $s_g$  and  $R_s$ . The coupling is achieved with the following algebraic equations:

$$P_c = p_g - p_w \quad (7)$$

$$s_w + s_g = 1.0 \quad (8)$$

$$R_s = f(p_w) \quad (9)$$

The capillary pressure  $P_c$  is defined as the difference between the non-wetting and the wetting phase pressures. Solubility of gas in water is assumed to be a function of water pressure only.

The non-linear coefficients for relative permeability ( $k_{rw}$  and  $k_{rg}$ ) and capillary pressure ( $P_c$ ) are evaluated as unique functions of saturations ( $s_w$  and  $s_g$ ).

## 2.2 FINITE DIFFERENCE APPROXIMATIONS

The finite difference solution of the system of partial differential equations is achieved by:

- discretization of spatial domain of interest into grid blocks and, if transient, the time domain into time increments,
- application of boundary conditions,
- establishment of initial conditions (if transient solution),
- development of finite difference approximations within each grid block over time, and
- matrix solution of finite difference equations.

The equivalent difference equations are conceptually derived by obtaining a balance for gas and water over each finite difference grid block.

Substituting Equations (7) and (8) in Equations (5) and (6) to eliminate  $p_w$  and  $s_w$ , and writing the finite difference approximations of the resulting equations, we obtain the following:

$$\Delta [T_w \Delta(p_g - P_c - \rho_w g h)] - Q_w = V_b \frac{\delta}{\Delta t} [\phi \rho_w (i - s_g)] \quad (10)$$

$$\Delta [R_s T_w \Delta (p_g - P_c - \rho_w g h) + T_g \Delta (p_g - \rho_g g h)] - Q_g - R_s Q_w = V_b \frac{\delta}{\Delta t} [R_s \phi \rho_w (1 - s_g) + \phi \rho_g s_g] \quad (11)$$

In the above equations:

$Q_w, Q_g$  = block volumetric recharge/withdrawal rate for water and gas phases,

$T_w, T_g$  = block edge transmissibilities for water and gas phases,

$\Delta$  = spatial difference operator defined for a variable  $\psi$  as  
 $\Delta_{i+1/2} \psi = \psi_i - \psi_{i+1}$

$\delta$  = temporal difference operator defined for a variable  $\psi$  as  
 $\delta \psi = \psi^{n+1} - \psi^n$

$\Delta t$  = time step size

and the block bulk volume,  $V_b$  is defined as:

$V_b = \Delta x \Delta y \Delta z$  for Cartesian geometry,

or (12)

$V_b = \frac{4}{3} \pi \Delta r^3$  for spherical geometry,

where  $\Delta x, \Delta y, \Delta z$  are the dimensions of the block and  $\Delta r$  is the radius.

Note that all terms above should be subscripted  $i, j, k$  to represent the coordinate location of the block in three-dimensional space (e.g.,

$V_b = V_{b;i,j,k}$ ;  $\phi = \phi_{i,j,k}$ ;  $P_w = P_{w;i,j,k}$ ;  $\Delta x = \Delta x_{i,j,k}$ ; etc.). An example of the spatial difference operator  $\Delta$  is given by:

$$\begin{aligned} \Delta [T \Delta (p_w^{n+1})] &= \Delta_x [T_{wx} \Delta_x (p_w^{n+1})] + \Delta_y [T_{wy} \Delta_y (p_w^{n+1})] \\ &+ \Delta_z [T_{wz} \Delta_z (p_w^{n+1})] \end{aligned} \quad (13)$$

with its first term defined by:

$$\begin{aligned} \Delta_x [T_{wx} \Delta_x (p_w)]_{i,j,k} &= T_{w;i-1/2,j,k} (p_{w;i-1,j,k} - p_{w;i,j,k}) \\ &- T_{w;i+1/2,j,k} (p_{w;i,j,k} - p_{w;i+1,j,k}) \end{aligned} \quad (14)$$

The finite difference transmissibility parameter  $T$  is defined as, for example:

$$T_{w;i+1/2,j,k} = \left( \frac{\Delta y \Delta z}{\Delta x} \right)_{i+1/2,j,k} \left( \frac{k_{xx}^k}{\mu_w^k} \frac{r_w}{w} \right)_{i+1/2,j,k} \quad (15)$$

where  $k_{xx}$  represents the x-direction component of the permeability tensor. The values of these variables in blocks  $i,j,k$  and  $i+1,j,k$  are weighted. A harmonic average value is used for the permeability, whereas a backward difference approximation in space is used for relative permeabilities.

The model is designed to be fully implicit, which means that all the quantities are expressed at the new time level. This is done by linearizing the non-linear parts of the partial differential equations by using an iterative Newton-Raphson scheme to predict the capillary pressures and relative permeabilities at the new time level.

### 2.3 FLUID AND ROCK PROPERTIES

The model assumes that the liquid phase may contain dissolved gas. It is also assumed that fluids are at constant temperature and in thermodynamic equilibrium.

Fluid densities at the subsurface system conditions are related to densities at reference conditions. The accumulation terms for the water and gas flow equations may be written as follows:

Water -

$$\frac{\partial(\phi\rho_w s_w)}{\partial t} = s_w\rho_w \frac{\partial\phi}{\partial t} + s_w\phi \frac{\partial\rho_w}{\partial t} + \rho_w\phi \frac{\partial s_w}{\partial t} \quad (16)$$

Gas -

$$\begin{aligned} & \frac{\partial(\phi\rho_w s_w R_S + \phi\rho_g s_g)}{\partial t} \\ &= \rho_w s_w R_S \frac{\partial\phi}{\partial t} + s_w R_S \phi \frac{\partial\rho_w}{\partial t} + \phi\rho_w \frac{\partial(s_w R_S)}{\partial t} \\ & \quad + \rho_g s_g \frac{\partial\phi}{\partial t} + s_g \phi \frac{\partial\rho_g}{\partial t} + \rho_g \phi \frac{\partial s_g}{\partial t} \end{aligned} \quad (17)$$

Rock and water compressibilities can be assumed constant under most conditions. This corresponds to linear variation of porosity and water density.

The rock and water compressibilities can be defined as follows:

$$c_R = \frac{1}{\phi^0} \frac{\partial\phi}{\partial p} \quad (18)$$

and

$$c_w = \frac{1}{\rho_w^0} \frac{\partial \rho_w}{\partial p} \quad (19)$$

And using

$$\frac{\partial \phi}{\partial t} = \frac{\partial \phi}{\partial p} \cdot \frac{\partial p}{\partial t}$$

We have

$$\frac{\partial(\phi \rho_w s_w)}{\partial t} = (s_w \rho_w \phi^0 c_R + s_w \phi \rho_w^0 c_w) \frac{\partial p_w}{\partial t} + \rho_w \phi \frac{\partial s_w}{\partial t} \quad (20)$$

and

$$\begin{aligned} & \frac{\partial(\phi \rho_w s_w R_S + \phi \rho_g s_g)}{\partial t} \\ &= (\rho_w s_w R_S + \rho_g s_g) \cdot \phi^0 c_R \frac{\partial p_w}{\partial t} \\ & \quad + s_w R_S \phi \rho_w^0 c_w \frac{\partial p_w}{\partial t} + \phi \rho_w \frac{\partial(s_w R_S)}{\partial t} \\ & \quad + s_g \phi \frac{\partial \rho_g}{\partial p_g} \frac{\partial p_g}{\partial t} + \rho_g \phi \frac{\partial s_g}{\partial t} \end{aligned} \quad (21)$$

The density of gas need not be a linear function of pressure.

Fluid viscosity is assumed to be a function of pressure only.

## 2.4 MODEL ASSUMPTIONS

This two-phase flow code for environmental applications is based on the following assumptions:

- There are two immiscible fluid phases,
- The two fluid phases represent water and air, or water and another gas, or water and any immiscible liquid, such as an organic solvent,
- There may be one solid phase with primary porosity (porous medium) or with a dual porosity, primary and secondary (fractures and matrix) porosities.
- At least one fluid phase is present in each portion of the domain,
- The density and viscosity of the fluids are functions of fluid pressure only,
- Darcy's law applies,
- The intrinsic permeability tensor principal axes are oriented along the (Cartesian) coordinate system,
- Hysterisis of the relative permeability and capillary pressure curves may be included,
- Water is only slightly compressible with a constant compressibility coefficient,  $c_w$ ,
- Gas (air) is compressible with a pressure-dependent compressibility coefficient  $c_g(p_g)$ ; only one phase can be considered a gas; the gas phase can also be used to represent a second immiscible liquid,

- The soil matrix is slightly compressible,
- The soil properties can be completely heterogeneous with respect to the spatial distribution of permeabilities, porosities, and capillary pressure curves.

### 3.0 TECHNICAL APPROACH

This section provides an overview of the general ECLIPSE 100 features.

## ECLIPSE FEATURES

---

### ECLIPSE100 Features

---

#### GENERAL

ECLIPSE100 is a fully-implicit, three phase, three dimensional, general purpose black oil simulator with gas condensate options. The program is written in FORTRAN77 and operates on any computer with virtual storage and an ANSI standard FORTRAN77 compiler. Suitable computers include HP9000, VAX11/780, MV8000, CRAY, IBM3081 etc.. Ancillary programs included in the ECLIPSE100 package are;

- GRAF a device independent graphics post-processor,
- PSEUDO a three dimensional pseudo function generator,
- FILL a corner-point geometry pre-processor,
- VFP a well bore hydraulics pre-processor, and
- EDIT a screen editor specially designed for ECLIPSE data preparation.

#### FREE FORMAT INPUT

Input data for ECLIPSE is prepared in free format using a keyword system. Any standard screen editor may be used to prepare the input file. Alternatively a special ECLIPSE screen editor called EDIT may be used. EDIT checks data as it is input and its 'help' facility contains the entire ECLIPSE reference manual.

#### PHASE OPTIONS

ECLIPSE100 can be used to simulate 1,2 or 3 phase systems. Two phase options ( oil/water, oil/gas, gas/water ) are solved correctly as two component systems, saving both computer storage and computer time. In addition to gas dissolving in oil ( variable bubble point pressure or gas/oil ratio ), ECLIPSE may also be used to model oil vapourising in gas ( variable dew point pressure or oil/gas ratio ).

#### GEOMETRY OPTIONS

Both corner-point and conventional block-centre geometry options are available in ECLIPSE. Radial and Cartesian block-centre options are available in 1, 2 or 3 dimensions. A 3D radial option completes the circle allowing flow to take place across the 0/360 degree interface. (keyword COORDSYS)

The ECLIPSE corner-point geometry option is unique and allows extremely complex geometries to be constructed to give a faithful representation of the reservoir geology. A special program called FILL is used to prepare corner-point data for ECLIPSE. The Eclipse Graphics Package may be used to display the grid in a

## ECLIPSE FEATURES

variety of ways. For example, in a large 3D study, the user may request a number of XZ cross-sections to be displayed simultaneously. Transparent areal overlays of the grid are useful for precise checking against geological maps.

Corner-point geometry is especially useful for highly faulted reservoirs. The grid may be distorted areally to fit along fault lines and displaced vertically to model complex scissor faults. Even sloping faults may be specified easily and precisely using the FILL system. The resulting non-neighbour grid transmissibilities are computed automatically by ECLIPSE. Flow across displacement faults is handled efficiently by the ECLIPSE solution procedures.

Other uses of corner-geometry include stream tube modelling and local grid refinement.

### FULLY IMPLICIT TECHNOLOGY

ECLIPSE uses the fully-implicit method to provide stability over long time steps. Care is taken to ensure that the non-linear fully-implicit equations are solved precisely by reducing all residuals to very fine tolerances. Material balance errors ( residual sums ) are extremely small. Newton's method is used to solve the non-linear equations. The Jacobian matrix is fully expanded in all variables to ensure quadratic ( fast ) convergence. Various special methods are used to accelerate convergence in highly non-linear problems. The linear equations arising at each Newton iteration are solved simultaneously by Nested Factorization accelerated by Orthomin.

Traditionally, fully-implicit techniques are reserved for small coning studies, where many cell pore volumes of fluid may pass through small cells near the well bore during a single timestep. IMPES and semi-implicit techniques cannot be used on such problems unless the timesteps are reduced to impractically small values. When the fully-implicit method is used the resultant linear equations are strongly coupled and can only be solved efficiently by simultaneous methods and not by simple sequential methods. Small problems of this type can be solved efficiently by direct methods such as D4 Gaussian elimination. Iterative methods, required for large problems, such as SIP and LSOR, typically fail to converge. Thus most simulators cannot apply fully implicit methods to large problems. In ECLIPSE, these restrictions are removed by Nested Factorisation which solves large problems efficiently and reliably.

### IMPES

Although the Fully Implicit method is the standard solution procedure used by ECLIPSE it is sometimes advantageous to use the IMPES ( IMPLICIT Pressure EXPLICIT Saturation ) method, and this option is available in ECLIPSE. IMPES is potentially unstable and should only be used on studies with regular grid systems ( no small cells ) and small timesteps such as in history

## ECLIPSE FEATURES

matching. In ECLIPSE it is possible to use the IMPES and Fully Implicit methods at different times in the same run.

### NESTED FACTORISATION

The linear equations arising at each Newton iteration are solved using Nested Factorisation accelerated by Orthomin. This is the fastest iterative technique yet devised for the solution of large sets of sparse linear equations. The technique conserves material precisely at each iteration. Material balance errors can therefore arise only as a result of non-linearities. Off band matrix elements arising from non-neighbour connections are included in the factorisation procedure and are therefore handled with extreme efficiency. Computing costs increase only marginally faster than the number of active cells in the reservoir ( $N^{5/4}$ ) in contrast with direct methods ( $N^3$ ). Nested Factorisation is therefore particularly well suited to the solution of large problems. Two and three phase problems are solved simultaneously. Further details of the method may be found in the appendix on the Solution of the Linear Equations.

At each iteration, a new 'search direction' ( approximate solution ) is computed by Nested Factorisation. The Orthomin procedure, which minimises the sum of the squares of the residuals ( errors ), requires each new search direction to be orthogonalised to the previous search directions. Thus previous search directions must be stored, using up valuable computing memory. Fortunately it is usually sufficient to store only the last few search directions. The number of search directions is called NSTACK and this may be set by the user in the RUNSPEC data section. The default value of NSTACK is 10. A memory management option is available for problems requiring a large stack on computers with limited memory. This is activated by setting NSTACK = 0. The stack, which is unlimited in length, is then stored on disk and read back by the solver as required. The memory management option saves memory at the expense of increased clock time due to relatively slow disk to memory data transfer rates.

A compromise between the two extremes of holding the stack entirely in memory or entirely on disk is activated by setting NSTACK negative. For example, if NSTACK is set to -9 then the first 8 search directions are held in memory and the remainder are stored on disk. Because ECLIPSE computes the memory required at the start of each run, the user can experiment to find the best value of NSTACK for his particular problem.

The solver is highly vectorised due to the use of active cell addressing. For example, all the Orthomin calculations on the stack are completely vectorised. Matrix multiplications are vectorised using the vector run technique. However, the tridiagonal inversion at the heart of Nested Factorisation, is inherently recursive and cannot be vectorised.

## ECLIPSE FEATURES

### NON-NEIGHBOUR CONNECTIONS

In conventional grid systems each cell has only one neighbouring cell in each direction ( two in each dimension ). Thus in 2D grids, each cell can have up to four neighbours and in 3D grids each cell can have up to six neighbours. Flow only takes place between neighbouring cells.

In ECLIPSE, it is possible to connect selected pairs of non-neighbouring cells, allowing fluid to flow directly from one cell of the pair to the other. Typical applications of non-neighbour connections are:-

1. Completing the circle in 3D radial studies
2. Displacement faults allowing flow between different layers across the fault.
3. Local grid refinement allowing fine definition near wells, including the ability to include coning grids within a cartesian system.

If FILL is used to generate the grid then the non-neighbour transmissibilities arising in the first two applications above are computed automatically by ECLIPSE. Local grid refinement can be performed using a general non-neighbour connection facility. Examples of how to construct locally refined grids are provided later in the manual.

Non-neighbour connections give rise to off-band elements in the Jacobian matrix and these are included in the Nested Factorisation procedure used to solve the linear equations.

### RUN-TIME DIMENSIONING

All of the internal arrays in ECLIPSE are dimensioned at run time to minimise the use of computer memory. For example, each element of the Jacobian is a 2\*2 matrix in 2 phase runs and a 3\*3 matrix in three phase runs. A two dimensional problem gives rise to a Jacobian matrix with 5 bands, while a 3 dimensional problem gives rise to a Jacobian matrix with 7 bands. Clearly it would be inefficient to dimension the internal arrays to cover the worst possible case. Most of the data required for ECLIPSE to compute the size of its internal arrays is provided in the RUNSPEC data section. Some additional information ( relating to automatically generated non-neighbour connections, inactive cells, vertical equilibrium options etc. ) is deduced from data provided in the GRID section. Space is saved by not storing unnecessary data for inactive cells. The total memory required for each run is printed out before the simulation begins.

### VERTICAL EQUILIBRIUM

ECLIPSE contains a unique Vertical Equilibrium option for modelling segregated flow in three dimensions. The option may be used with either block-centre or corner-point geometry. With corner-point geometry, it takes account of the shape and orientation of each distorted grid block. A mixing parameter

## ECLIPSE FEATURES

enables the user to specify the degree of fluid segregation. At one extreme, total fluid segregation is assumed, while at the other extreme the fluids are assumed to be totally dispersed throughout each grid block. The VE facility takes account of hysteresis effects arising, for example, when oil invades the water zone and retreats, leaving a residual critical oil saturation.

### DUAL POROSITY

The ECLIPSE dual porosity/permeability option is suitable for simulating highly fractured reservoirs. Each grid block is represented by a matrix cell and a fracture cell. In the dual porosity option, flow through the reservoir takes place via the fracture cells only. Flow also takes place between each matrix cell and its corresponding fracture cell. Flow does not take place between neighbouring matrix cells. In the dual porosity/permeability option flow may also take place between neighbouring matrix cells. Gravity imbibition/drainage between matrix and fracture cells is accounted for.

### PVT AND ROCK DATA

ECLIPSE honours pressure and saturation function data precisely as it is specified by the user. It does not follow the practice, common in the industry, of smoothing the data by interpolating to a fixed number of equally spaced saturation intervals.

Different table numbers can be used for different parts of the reservoir. Thus, for example, separate saturation tables can be entered for each type of rock.

### DIRECTIONAL RELATIVE PERMEABILITIES

Relative permeability curves may be specified separately for flow in the horizontal and vertical directions. The user has the option of specifying 3 (X,Y,Z) or 6 (+X,-X,+Y,-Y,+Z,-Z) relative permeability curves for each saturation function region, rather than the usual single curve. This additional degree of freedom enables the engineer to improve the physics of a simulation study in a relatively simple manner. For example, most reservoir cells have small vertical dimensions and large areal dimensions, and horizontal flow will be more realistically modelled if the horizontal relative permeability is small below the Buckley-Leverett saturation. Directional relative permeabilities are also required for the ECLIPSE pseudo option which may be used on a three dimensional coarse grid.

## ECLIPSE FEATURES

### SATURATION TABLE SCALING

The connate, critical and maximum saturation endpoints of the relative permeability and capillary pressure curves may be input separately for each grid cell within the reservoir or, alternatively, as a series of depth tables for regions within the grid. The scaling option allows the user to specify relative permeability and capillary pressure data which are functions of a normalised saturation. In addition, it allows the modelling of reservoirs where depth variations occur in the initial critical or connate fluid saturations. The user may specify that the saturation table scaling is to be isotropic, distinguish between coordinate lines (X,Y,Z) or distinguish between coordinate directions (+X,-X,+Y,-Y,+Z,-Z) .

The saturation table scaling facility may be used to equilibrate a reservoir model with the correct initial quantities of mobile fluid-in-place by means of a special correction applied at the fluid contacts.

### ROCK COMPACTION

The Rock Compaction option models the collapse of pore channels as the fluid pressure decreases. The process may be reversible or not as required. The facility is particularly useful in dual porosity runs.

### TRACER TRACKING

The tracer tracking option is a general facility to determine the movement of 'marked' fluid elements during a simulation run. It may be used, for example, to differentiate the movement of water injected by different wells and initial aquifer water, or to predict the variation in salinity or concentrations of other chemical species.

Tracers may also be defined to exist in a hydrocarbon phase, and used to model the exchange of the fluid elements between the free and solution states whenever mass transfer takes place in the associated hydrocarbon phase. The tracer option is therefore useful in predicting the movement of the initial free and/or solution gas. Initial sulphur content may be defined as a function of depth and tracked with the movement of oil.

An option to control the effects of numerical diffusion on tracer interfaces is available.

### API TRACKING

Unlike the 'passive' tracer tracking options described above, API tracking is 'active'. It takes full account of the mixing of oils with totally different

## ECLIPSE FEATURES

PVT properties. This contrasts with the conventional practice of assuming that the oil magically assumes the formation volume factor, viscosity, density and bubble point of the cell into which it flows. Initial API may be defined as a function of depth in each equilibration region or independently for each cell. A 4th equation is solved at each timestep to update APIs.

### HYSTERESIS

Both relative permeability and capillary pressure hysteresis effects can be modelled. There is a choice of two models for relative permeability hysteresis in the non-wetting phase: Carlson's model and Killough's model. There is also an option of using Killough's wetting phase hysteresis model in two-phase runs.

### INDIVIDUAL WELL CONTROLS

ECLIPSE has a comprehensive set of options for controlling individual wells. Producers can operate at a specified value of the oil rate, water rate, liquid rate, gas rate, reservoir fluid voidage rate, bottom hole pressure and tubing head pressure. The engineer supplies a target value for one of these quantities, and limiting values for any of the remaining quantities. The well will operate at its specified target as long as none of the limits are violated. If a limit is going to be violated, the well will automatically change its mode of control to keep operating within its allowed limits. Efficiency factors can be applied to take account of regular downtime. The flow rates and pressures will be calculated for the well under normal flowing conditions, but the cumulative flows will be reduced according to the efficiency factor. Injection wells have a similar set of controls. Target or limiting values can be supplied for the injection rate at surface conditions, the injection rate at reservoir conditions, the bottom hole pressure and the tubing head pressure.

Production wells can also be subject to an additional class of 'economic' constraints. A producer will automatically be shut-in if its oil or gas production rate falls below an economic limit. If its water cut, gas-oil ratio or water-gas ratio exceeds the specified upper limit, the well's worst-offending completion will be closed, or the well itself will be shut-in. Water cut, gas-oil ratio and water-gas ratio limits can also be applied to individual connections, which will be closed if their limits are violated. In addition there is an option to limit the water cut, gas-oil ratio or water-gas ratio from a well by successively reducing its production rate each time the limit is violated. Wells subject to a minimum tubing head pressure limit can have their productivity boosted by automatically switching on artificial lift when their oil or liquid flow rates fall below a specified limit, subject to constraints on the group or field capacity for artificial lift. Individual wells can be 'flagged' to make the simulation run stop if they are shut-in for any reason. The run can also be made to stop if all the producers in the

## ECLIPSE FEATURES

field, or in one or more nominated groups, are shut-in. Wells that are shut-in can be tested periodically to see if they can flow economically again.

### GROUP AND FIELD PRODUCTION CONTROLS

The simulator contains facilities to control the collective behaviour of groups of wells, or the field as a whole. The overall production rate of oil, water, gas or liquid from one or more groups can be made to meet a specified target. The target rate is apportioned between the various producers in proportion to either their production potentials or specified 'guide rates', but ensuring that no well violates its individual flow rate or pressure limits. When the group no longer has enough potential to meet its production target, the production rate will decline. However, the simulator can postpone the decline by automatically opening new wells from a drilling queue whenever extra potential is required, subject to constraints on the drilling rate and the maximum number of open wells allowed in each group.

Upper limits can also be applied to groups' production rates of oil, water, gas and liquid. There is a choice of actions that will be taken whenever one of these limits is violated, ranging from a work-over of the worst-offending well to the application of group control to hold the group's production rate at the limiting value. In addition, a set of 'economic constraints' equivalent to those described above for individual wells can be applied to the overall production behaviour of groups.

The same set of controls and limits can also be applied to the overall field production. A target production rate for the field can either be apportioned between the groups in proportion to specified group guide rates, or apportioned directly between the wells in proportion to their production potentials or guide rates. The field production rate can be controlled simultaneously with the production from one or more of its constituent groups. The identity of the controlled phase (oil, water, gas or liquid) need not be the same in each case. For example, the field can produce oil at a required rate while one of the groups is producing at its maximum gas handling capacity. Thus the field will produce at its target rate while each group honours its own set of constraints, and while each well honours its individual pressure and rate constraints.

### MULTI-LEVEL GROUPING HIERARCHY

Groups can be arranged into a multi-level tree structure of any required depth, by declaring some groups to be 'sons' of higher level groups. All group controls and limits can be applied at any level (including the field, which can be regarded as a group at level zero at the top of the tree).

## ECLIPSE FEATURES

### GROUP INJECTION CONTROLS

Groups, and the field, can be given targets and limits to control their injection rates. The engineer can set target or limiting values for any of the following quantities:

- (1) surface injection rate of a particular phase
- (2) total reservoir volume injection rate of all phases
- (3) re-injection fraction of a particular phase
- (4) total voidage replacement fraction by all injected phases.

Thus for example a group can be made to re-inject a set proportion of its produced gas, and to inject water to make up an overall reservoir volume injection target or voidage replacement fraction.

A group injection target is apportioned between the wells in proportion to either their injection potentials or specified guide rates, subject to the wells' individual flow and pressure limits. A target for a high level group (or the field) can optionally be apportioned among its subordinate groups in proportion to their group guide rates, which can be set directly or calculated to equal their production voidage rates or voidage replacement needs. If a group (or the field) has insufficient injection potential to meet its target, the drilling queue will be scanned for a suitable injector to open.

### SALES GAS PRODUCTION CONTROL

The sales gas production rate from any group or the whole field can be controlled to meet a specified target. The sales gas production rate of a group or the field is defined as its total gas production rate, minus its total gas injection rate, minus the gas consumption rate of the group and any subordinate groups. (The group gas consumption rates can be set individually.)

The sales gas production rate is controlled by re-injecting the surplus gas not required for sale. Sales gas control can be applied independently of any other production controls on the group or field, provided that there is enough injection capacity to inject the surplus gas. The group or field is automatically placed under gas re-injection control, and its target re-injection fraction is determined dynamically at each time step to inject the surplus gas.

### CROSSFLOW AND CO-MINGLING IN WELLS

When a well is completed in more than one grid block, the flow rate from each grid block is proportional to the product of three quantities:

## ECLIPSE FEATURES

- (1) the transmissibility between the grid block and the well bore
- (2) the mobility of the phase in the grid block at the perforations
- (3) the pressure drawdown between the grid block and the well bore.

ECLIPSE takes full account of all three quantities when apportioning a well's target flow rate between its various grid block connections. This is especially important in cases where a well is completed in two or more layers of a reservoir with poor vertical communication, where the drawdown at each layer can be substantially different.

In certain situations the drawdown in one layer may have the opposite sign to the drawdown in the other completed layers. The simulator will then allow crossflow to take place between the reservoir layers through the well bore. A material balance constraint is observed within the well bore at formation level, so that the mixture of phases flowing out of the well bore corresponds to the average phase mixture entering the well bore. ECLIPSE can handle crossflow in any type of well operating under any mode of control, including the case when the well is plugged off above the formation while the completions remain open to the reservoir. The crossflow facility can be turned off if the engineer does not wish this to occur.

### HIGHLY DEVIATED AND HORIZONTAL WELLS

There is no restriction on the location of the completions within each well. A highly deviated well can be completed in a number of grid blocks that do not all belong to the same vertical column. In fact a well can be completed in several grid blocks within the same layer, enabling the simulator to model wells completed in horizontally-adjacent grid blocks. This feature is also useful in 3-D radial coning studies where the well connects with all the innermost, wedge-shaped, blocks in any given layer.

### SPECIAL FACILITIES FOR GAS WELLS

ECLIPSE has two facilities for improving the accuracy of modelling gas inflow. Firstly, the non-Darcy component of the pressure drawdown resulting from turbulence near the well bore is modelled by a flow-dependent contribution to the skin factor. The simulator calculates the turbulence skin factor and the gas inflow rate in each completion by a method which ensures that they are fully consistent with each other. The calculation is performed fully implicitly, to maintain stability over long time steps. These precautions are especially important when the pressure drop due to turbulence is a significant proportion of the total drawdown.

The second facility is the option to use either the Russell Goodrich equation or the gas pseudo-pressure equation in place of the conventional inflow performance relationship for free gas flow. Both equations make allowance for the variation in the viscosity and the density of the gas over the

## ECLIPSE FEATURES

pressure drop between the grid block and the well bore. For the Russell Goodrich equation, these quantities are determined at the average pressure between the grid block and the well bore, instead of at the pressure of the grid block. For the pseudo-pressure equation, these quantities are replaced by an average value integrated between the grid block and well bore pressures.

### FINE GRID EQUILIBRATION

The initial pressures and saturations in the reservoir grid blocks can be calculated by the 'equilibration' facility. Each phase in the reservoir is given a hydrostatic pressure gradient, dependent on its local density. The saturations are determined from the local pressure differences between the phases, using the capillary pressure tables.

A common practice is to set the fluid saturations within each grid block equal to the local saturations at the centre of the block. However, this procedure can produce significant errors in the Fluids-in-Place estimation when a large grid block contains a fluid contact or part of a transition zone. ECLIPSE has an option to improve the accuracy of the Fluids-in-Place calculation in this situation. The fluid saturations are determined at a number of levels within the block, and these are averaged to obtain the overall fluid saturations within the block. The method can also take account of the shape of sloping grid blocks, by weighting the fluid saturations at each level within the block with the area of the horizontal cross-section of the block at that level.

### AQUIFERS

Aquifers may be represented by a choice of two analytic models and a numerical model. The analytic models are the Fetkovich aquifer and the Carter-Tracy aquifer. The numerical model consists of a sequence of aquifer cells connected together in one dimension. The innermost cell of the numerical aquifer sequence may be connected to any number of reservoir cells. The depth, dimensions, porosity, permeability etc. of each aquifer cell may be specified separately, giving the user complete flexibility in defining each aquifer.

### ANCILLARY PROGRAMS

Ancillary programs included as part of the ECLIPSE system are described below.

FILL generates unconventional grid systems based on corner point geometry. Grids may be distorted to fit along fault lines. Sloping displacement faults, complex scissor faults, distorted wedge shaped grids etc. may be generated easily using FILL. As its name implies, FILL is specially designed to interpolate sparse data which may only be known at a few points on the grid. Missing values are 'filled' in to generate complete

## ECLIPSE FEATURES

grid data files for ECLIPSE. For example, rock properties such as porosity need only be specified at a few points in each grid layer and FILL will provide the rest.

VFP calculates pressure drops in wells and pipelines, and generates vertical flow performance tables. These are multi-dimensional tables relating a well's bottom hole pressure to the flow rate, tubing head pressure, water fraction and gas fraction. The tables must be supplied as input data to ECLIPSE when the simulation involves tubing head pressure constraints. ECLIPSE performs calculations involving tubing head pressure by interpolating these tables. This method is more efficient than performing the actual pressure drop calculations within ECLIPSE, which would have to be done iteratively several times each time step. The table look-up procedure also makes it easier to perform the tubing head pressure calculations fully implicitly in all variables.

VFP performs a pressure traverse calculation for specified combinations of flowing conditions, using a choice of standard multi-phase flow correlations. The calculations can include the effects of gas lift, a down-hole pump, a gas compressor, and a surface choke.

There is a choice of four multi-phase flow correlations:

- (1) Aziz, Govier and Fogarasi,
- (2) Orkiszewski,
- (3) Hagedorn and Brown,
- (4) Beggs and Brill.

Different correlations may be used for different sections of tubing, enabling VFP to handle horizontal and undulating pipelines as well as vertical well bores and risers.

EDIT provides an interactive full screen editor specially designed to simplify the construction of ECLIPSE input data files. At the top level EDIT shows an overview of the input file, displaying keywords but not their attendant data. It is then possible to zoom into a selected keyword, and EDIT will display the appropriate data form. For example, the data form for keyword SWFN ( Water Saturation Functions ) consists of a number of tables each containing three columns ( water saturation, relative permeability of water, water-oil capillary pressure ). The different tables apply to different reservoir regions or, if the directional relative permeability option is used, to different flow directions. As data is entered into the tables, EDIT performs simple data checks and warns the user of errors. A help line, at the top of the screen, contains useful information about each data item. On line access to the ECLIPSE manual may be obtained by entering a HELP command displayed at the top or bottom of the screen, where several other 'menu' options are also displayed. If the HELP key is pressed while editing the SWFN keyword, the appropriate page of the manual describing SWFN is displayed on the screen.

## ECLIPSE FEATURES

GRAF provides an easy to use, menu driven graphics capability. Typical uses of GRAF are:

1. Line graphics showing, for example, field pressure against time.
2. Grid graphics showing the reservoir grid including faults.
3. Dynamic graphics showing animated colour time displays of saturations or pressures etc.

Line graphics are generated from 'SUMMARY' files created by ECLIPSE at each report time. Each summary file contains data relating to field, group, well, connection, region and cell quantities recorded at intermediate timesteps. Thus the summary files ( like the restart files ) may be used to monitor runs in progress.

Grid graphics are generated from a 'GRID' file created by ECLIPSE at the start of a simulation run. Grid files contain the locations of the corners of each cell in the reservoir and may be used by the graphics package to display the grid in a variety of ways.

Dynamic graphics are generated from 'RESTART' files created by ECLIPSE at prescribed times. Restart files contain detailed information such as saturations and pressures for each grid cell. Their main purpose is to enable ECLIPSE to be restarted at the prescribed times. However, they may also be used in conjunction with the grid file to generate colour displays of saturations and pressures. The graphics package enables the user to step forwards or backwards through colour sequences creating animated displays of water floods, gas coning events etc.

PSEUDO is the Eclipse Pseudo Package for generating coarse grid pseudo functions for low-cost sensitivity studies. PSEUDO uses Eclipse restart files, generated during a fine grid simulation, to generate pseudos for a coarse grid model. The fine grid model may be a cross-section with many layers or a large 3D model. The coarse grid model may be a 1D, 2D or 3D model and is formed by amalgamating neighbouring cells of the fine grid model into large coarse grid cells. Because PSEUDO runs separately from ECLIPSE, it is possible to redefine the coarse grid and generate the new pseudos inexpensively. Directional pseudo relative permeabilities, generated by PSEUDO and appropriate to each face of the coarse grid, may be input to the ECLIPSE coarse grid model to reproduce the results of the fine grid model or to perform low-cost sensitivity studies.

## PASSWORDS

Some companies purchase or rent a restricted version of ECLIPSE with various facilities removed. Restricted versions of ECLIPSE100 will only run with special passwords provided by ECL. The restricted range of facilities may be extended on request with the issue of a new password from ECL.

#### 4.0 SOLUTION OF FINITE-DIFFERENCE EQUATIONS

This section describes solution of the finite-difference equations. This section includes solution of the linear equations, description of the fully implicit and the IMPES (semi-implicit - Implicit Pressure, Explicit Saturation) methods, general transmissibility calculations for the block averaged and corner-point geometries.



## SOLUTION OF THE LINEAR EQUATIONS

It should rarely be necessary to alter the default settings of NSTACK and LITMAX. However, for very difficult problems, they may have to be increased to ensure that the linear equations are solved correctly. Warning messages are printed by ECLIPSE when the linear equations are not fully converged.

The total memory required for each run is printed out by ECLIPSE as soon as the required data has been read in and analysed.

On non-virtual machines with limited memory, such as the Cray, it may be necessary to set NSTACK = 0. This initiates the automatic memory management system which works by storing all previous search directions on disk. Computer memory requirements per cell are reduced but elapsed time is increased due to increased I/O. Disk I/O charges are high on service bureau computers so it pays to minimise disk usage by setting the maximum value of NSTACK permitted within the memory available. This may not always be possible especially when running very large problems. The extended memory management option, obtained by setting NSTACK negative, may be useful in such cases. For example, if NSTACK = -5, then the first 4 search directions are retained in memory and the remaining search directions are stored on disk if they are required.

### 2. Material Balance

At each Newton iteration we solve the linear equation

$$A.x = b$$

where A is the Jacobian matrix ( $dR/dX$ ), and b is the non-linear residual of the prior Newton iteration,  $b = R = dM/dt + F + Q$ . If the elements of b are summed over all cells in the reservoir then the flow terms, F, will cancel and the sum corresponds to the rate of mass accumulation in the reservoir. ECLIPSE computes an initial approximate solution, y, by solving an approximate equation

$$B.y = b$$

The material balance error ( i.e. the error in the mass accumulation rate ) corresponding to y is obtained by summing the elements of the residual, r

$$r = b - A.y = (B - A).y \quad \text{sum} = \sum_{i=1}^{\text{ncells}} (r_i)$$

This sum is zero if B is chosen such that

$$\text{colsum}(B) = \text{colsum}(A)$$

where colsum(A) is the diagonal matrix formed by summing A in columns. The

## SOLUTION OF THE LINEAR EQUATIONS

above colsum constraint is obeyed independently for each component ( oil, water and gas ) by ECLIPSE.

### 3. Nested Factorization

Three dimensional finite difference systems give rise to sparse linear equations with banded matrices (Fig 1 ) such as

$$A = D + L1 + U1 + L2 + U2 + L3 + U3$$

where

D is a diagonal matrix

L1,L2 and L3 are lower bands

U1,U2 and U3 are upper bands

The elements of D,L1,L2,L3,U1,U2,U3 are 2\*2 matrices for 2-phase systems (eg oil/water, oil/gas, gas/water ) and 3\*3 matrices for 3-phase systems. L2 and U2 connect cells in direction 2 etc.

In Nested Factorization, an approximation, B, is constructed by the following nested sequence of factorizations

$$\begin{aligned} B &= ( P + L3 ).PINV.( P + U3 ) = P + L3 + U3 + L3.PINV.U3 \\ P &= ( T + L2 ).TINV.( T + U2 ) = T + L2 + U2 + L2.TINV.U2 \\ T &= ( G + L1 ).GINV.( G + U1 ) = G + L1 + U1 + L1.GINV.U1 \end{aligned}$$

where G is a diagonal matrix and PINV is the inverse of P etc. It follows that

$$\begin{aligned} B &= G + L1 + U1 + L2 + U2 + L3 + U3 \\ &\quad + L1.GINV.U1 + L2.TINV.U2 + L3.PINV.U3 \\ &= A + G - D + L1.GINV.U1 + L2.TINV.U2 + L3.PINV.U3 \end{aligned}$$

The diagonal matrix G is computed using

$$0 = G - D + L1.GINV.U1 + \text{Colsum}( L2.TINV.U2 + L3.PINV.U3 )$$

This choice of G ensures that  $\text{Colsum}(B) = \text{Colsum}(A)$  which, in turn, ensures that there are no material balance errors in the solution of the linear equations.

Off band elements, corresponding to non-neighbour connections, arising in the treatment of faults, local grid refinement, completing the circle in 3D radial studies etc., destroy the simple structure of the bands L1,U1..., but are easily incorporated in the Nested Factorization procedure by simply generalising the definition of the bands.

## SOLUTION OF THE LINEAR EQUATIONS

### 4. Initialisation

---

Before iteration begins we must compute the diagonal matrix G. ECLIPSE computes and stores GINV rather than G as this is more efficient in subsequent calculations. To conserve material, we have

$$G = D - L1.GINV.U1 - \text{colsum}( L2.TINV.U2 ) - \text{colsum}( L3.PINV.U3 )$$

The calculation proceeds one cell at a time; thus when GINV is known for a cell, we can calculate the contribution L1.GINV.U1 to the value of G in the next cell. When GINV is known on a line, we can calculate the contribution of colsum( L2.TINV.U2 ) to G on the next line. When G is known on a plane, we can calculate the contribution of colsum( L3.PINV.U3 ) to G on the next plane. The cost of computing G (and GINV) is reported under the mnemonic GINV in the ECLIPSE timing reports.

### 5. Solution Procedure

---

To compute a new search direction ECLIPSE solves the equation  $B.y = r$  by the following hierarchical procedure. At the outermost level we solve

$$\begin{aligned} & ( P + L3 ).( I + PINV.U3 ).y = r \\ \text{using (a)} & \dots\dots\dots y = PINV.( r - L3.y ) \\ \text{and (b)} & \dots\dots\dots y = y - PINV.U3.y \end{aligned}$$

Equation (a) is solved one plane at a time, starting with the first plane and progressing forward until y is known on each plane. The equation is explicit because L3.y involves the 'known' solution, y, on the previous plane. Similar considerations apply to the solution of equation (b) which is performed in reverse order starting with the last plane and sweeping backwards one plane at a time.

During the solution of equations (a) and (b) we must compute vectors of the form  $z = PINV.q$  on each plane. This involves solving equations like

$$\begin{aligned} & ( T + L2 ).( I + TINV.U2 ).z = q \\ \text{using (c)} & \dots\dots\dots z = TINV.( q - L2.z ) \\ \text{and (d)} & \dots\dots\dots z = z - TINV.U2.z \end{aligned}$$

Equation (c) is solved in a forward sweep through the plane, one line at a time. Similarly equation (d) is solved by sweeping backwards one line at a time.

Finally, we note that during the solution of equations (c) and (d) we must compute vectors of the form  $w = TINV.v$  on each plane. This involves solving the tridiagonal equation

## SOLUTION OF THE LINEAR EQUATIONS

$(G + L1).(I + GINV.U1).w = v$   
 using (e)..... $w = GINV.(v - L1.w)$   
 and (f)..... $w = w - GINV.U1.w$

Equations (e) and (f) are solved by sweeping first forwards, then backwards through the cells in a line.

Computing time required to calculate a new search direction is reported under the mnemonic SRCH.

### 6. Orthomin

The Orthomin procedure use by ECLIPSE is outlined below.

- |   |                        |         |
|---|------------------------|---------|
| (a) Compute the initial solution,   | $B.y = b$              | $x = y$ |
| (b) Compute the initial residual, r.  | $r = b - A.y$          |         |
| The sum of the elements of r is now zero ( due to colsum ).   |                        |         |
| (c) Test for convergence and escape if converged.   |                        |         |
| Otherwise ;   |                        |         |
| (d) Find a new search direction, y.   | $B.y = r$              |         |
| (e) Orthogonalise y to the stack of previous search directions (z).   | $y = y - (y.z)/(z.z)z$ |         |
| (f) Find the optimum step length, s, which minimises the saturation normalised sum of the squares of the residuals (r.r). | $q = A.y$              |         |
|   | $g = q.dt.Bave/PV$     |         |
| (g) Update the solution residual and stack.   | $s = (r.g)/(g.g)$      |         |
|   | $x = x + sy$           |         |
|   | $r = r - sq$           |         |
|   | $z = (z,y)$            |         |
- Go to (c).

### 7. Convergence

Convergence of the linear solver can be monitored by setting the 7th parameter of keyword DEBUG greater than zero. If set to 1, a debug report is generated showing, at each iteration for each phase;

- (a) saturation normalised root mean square residuals,
- (b) residual sums ( material balance errors ) and
- (c) absolute maximum saturation normalised residuals.

The orthomin procedure is set to minimise item (a) above. Item (b) should be zero to within rounding error, while item (c) is used to test for convergence. The standard convergence criterion is that the maximum saturation normalised

## SOLUTION OF THE LINEAR EQUATIONS

residual,  $|r|_{\max}$ , be less than TRGLCV (-0.0001). The value of TRGLCV may be reset using keyword TUNING, but this is seldom necessary and should not be done without good reason. Convergence is also assumed if  $|r|_{\max}$  is less than  $\text{Min}(0.1*|b|_{\max}, 0.001*r_{\text{top}}, \text{XXXLCV})$ . The default setting of XXXLCV is 0.001 and this may also be reset in TUNING.  $|b|_{\max}$  is the largest saturation normalised input residual and  $r_{\text{top}}$  is the greatest saturation normalised residual to occur at any stage of the procedure.  $r_{\text{top}}$  may be much larger than  $|b|_{\max}$  because the colsum constraint makes the initial residuals quite large. Values of the step length,  $s$ , and the orthogonalisation coefficients (beta) are also printed in the debug output. The perfect step is  $s = 1.0$ , but this is seldom achieved. If the 7th DEBUG parameter is set to 2 then the sum of the absolute un-normalised residuals is printed corresponding to the input residual,  $b$ , and the final residual,  $r = b - A.x$ . This is the 'residual from solution check'. If the 7th DEBUG parameter is set to 3 then residual maps are printed for each phase at each iteration.

### 8. Treatment of Wells

The strongly coupled fully-implicit treatment of wells in ECLIPSE extends the linear equations to include the well variables,  $w$ , corresponding to the bottom hole flowing pressure, and well bore flowing fractions. Thus each well, like each cell, has three variables in the three phase case. Including the well terms, the linear equations now become

$$A.x + C.w = r$$

$$R.x + D.w = u$$

where,  $u$  is the well residuals and if there are  $N$  grid blocks and  $L$  wells,

$A$  is a banded  $N*N$  matrix ( fig. 1 ),  $C$  is a sparse  $N*L$  matrix,

$R$  is a sparse  $L*N$  matrix and finally  $D$  is a sparse  $L*L$  matrix.

Usually  $D$  is lower triangular and easily inverted, allowing the well terms to be eliminated. Thus the equation for  $x$  becomes

$$( A - C.DINV.R ).x = r - C.DINV.u$$

which is easily solved by nested factorisation with Orthomin.

Further details are described in ' Nested Factorization ' SPE12264 by John Appleyard and Ian Cheshire ( San Francisco 1983 ), ' Enhancements to the Strongly Coupled, Fully Implicit Well Model: Wellbore Crossflow Modeling and Collective Well Control' SPE12259 by Jon Holmes ( San Francisco 1983 ), 'An Efficient Fully Implicit Simulator' SPEJ, 1983, 23, 544 by Dave Ponting, Paul Naccache, Richard Pollard, Steve Walsh et al. Recent advances in Nested Factorization, giving improved efficiency in release 8711, are reported by Ian Cheshire and Richard Pollard in the proceedings of the Cambridge Conference on Mathematical Techniques for the Oil Industry, July 1987.

## FULLY IMPLICIT AND IMPES METHODS

-----  
The Fully Implicit and IMPES Methods.  
-----

ECLIPSE contains options for both the fully implicit and IMPES solution procedures. The fully implicit method is totally stable and may be used to solve 'difficult' problems such as coning studies. Although IMPES is potentially unstable it is less dispersive and sometimes faster than the fully implicit method, and may be used on 'easy' problems such as history matching applications where the timesteps are usually small. Both methods may be used in the same run by using keywords IMPES or IMPLICIT at appropriate places in the SCHEDULE section. The default solution procedure is IMPLICIT.

### The Fully Implicit Method

The fully implicit residual, R, for each component in each grid block at each timestep is

$$R = dM/dt + F + Q$$

where

dM is the mass, per unit surface density, accumulated during the current timestep, dt,

F is the net flow rate into neighbouring grid blocks,

Q is the net flow rate into wells during the timestep.

The mass accumulation and flows are evaluated in terms of the pressures, P, and saturations, S, at the end of the timestep, i.e. at time t+dt.

These are implied by the non-linear residual equation

$$R(X) = 0$$

where, for a 3 component system ( oil,water,gas ), the residual R and the solution, X, are 3 component vectors in each grid block;

$$R = \begin{bmatrix} |R_o| \\ |R_w| \\ |R_g| \end{bmatrix} \quad X = \begin{bmatrix} |P_o \\ |S_w \\ |S_g \text{ or } R_s \text{ or } R_v| \end{bmatrix}$$

and the Jacobian, dR/dX, takes the form

FULLY IMPLICIT AND IMPES METHODS

$$dR_i/dX_j = \begin{bmatrix} dR_o/dP_o & dR_o/dS_w & dR_o/dS_g \\ | & & | \\ | & & | \\ dR_w/dP_o & dR_w/dS_w & dR_w/dS_g \\ | & & | \\ | & & | \\ dR_g/dP_o & dR_g/dS_w & dR_g/dS_g \end{bmatrix} ij$$

The mass change during the timestep, dt, is

$$dM = M(t+dt) - M(t)$$

with

$$M = PV \begin{bmatrix} S_o/B_o + R_v S_g/B_g \\ | \\ S_w/B_w \\ | \\ S_g/B_g + R_s S_o/B_o \end{bmatrix}$$

where

- PV is the pore volume
- B<sub>o</sub> is the oil formation volume factor
- B<sub>w</sub> is the water formation volume factor
- B<sub>g</sub> is the gas formation volume factor
- R<sub>s</sub> is the solution gas/oil ratio
- R<sub>v</sub> is the vapour oil/gas ratio.

When S<sub>g</sub> is zero the solution variable becomes R<sub>s</sub> ( undersaturated oil ) and when S<sub>o</sub> is zero the solution variable becomes R<sub>v</sub> ( undersaturated gas ). Terms in the Jacobian are adjusted in accordance with the change of variable. No approximations are made in evaluating the Jacobian in ECLIPSE. Great care is taken to compute all derivatives to ensure quadratic convergence of Newton's method.

Hydrocarbon states are:

- State 1 Gas only, R<sub>s</sub> = 0, S<sub>g</sub> = 1 - S<sub>w</sub>, variables are P<sub>o</sub>, S<sub>w</sub>, R<sub>v</sub>.
- State 2 Gas & oil, R<sub>v</sub> = R<sub>v</sub>sat, R<sub>s</sub> = R<sub>s</sub>sat, variables are P<sub>o</sub>, S<sub>w</sub>, S<sub>g</sub>.
- State 3 Oil only, R<sub>v</sub> = 0, S<sub>g</sub> = 0, variables are P<sub>o</sub>, S<sub>w</sub>, R<sub>s</sub>.

The number of cells in each state, together with the number of cells changing state during a Newton iteration ( NTRAN ), are printed in the summary of each Newton iteration.

Newton's method is used to solve the non-linear residual equations R(X) = 0.

Thus at each iteration we :-

1. Solve the linear equation,  
 $R(X-x) \sim R(X) - (dR/dX).x = 0$  ( using Nested Factorization ).
2. Update the solution vector,  
 $X = X - x$
3. Recompute the residual, R(X).

## FULLY IMPLICIT AND IMPES METHODS

Several iterations may be required to reduce the residuals to a 'sufficiently small' value. In ECLIPSE, two measures are used to define the meaning of 'sufficiently small'. These are the material balance error and the maximum saturation normalised residual.

### The IMPES Method

The IMPES ( IMPLICIT Pressures EXPLICIT Saturations ) residual is similar to the fully implicit residual except that all flow and well terms are computed using saturations (or  $R_s, R_v$  ) at the beginning of each timestep.

$$R = (M(t+dt) - M(t))/dt + F(P(t+dt), S(t)) + Q(P(t+dt), S(t))$$

The mass terms  $M(t+dt)$  are evaluated using both pressures and saturations at the end of the timestep. This makes the non-linear residual equation,  $R = 0$  much easier to solve because there are now no non-linearities arising from relative permeabilities which remain fixed throughout the timestep. However, to solve the IMPES equations correctly it is still necessary to iterate until all residuals have been reduced to a sufficiently small value.

The linear equations arising from Newton's method are also much easier to solve in the IMPES case because derivatives of flows wrt saturations are zero. The linear equations are solved sequentially, first for pressure and subsequently for saturation changes. This contrasts with the fully implicit method where the linear equations must be solved simultaneously.

Timesteps are chosen automatically to limit saturation changes to 5%. However a converged timestep is accepted if the largest saturation change is less than 10% and if the largest pressure change is less than 200psi/atmos. These values may be altered using keyword IMPES.

### Material Balance

If the residuals are summed over all cells in the reservoir, the flow terms  $F$  will cancel, because the flow out of one cell is always equal and opposite in sign to the corresponding flow into its neighbouring cell. Thus the sum of the residuals for each phase or component corresponds to the net mass accumulation within the reservoir less the net influx through wells. This is the material balance error. For a 3 component system we have

$$\begin{aligned} \text{SUM}_i(R_o)_i &= \text{SUM}_i(dM_o/dt)_i + \text{SUM}_i(Q_o)_i \\ \text{SUM}_i(R_w)_i &= \text{SUM}_i(dM_w/dt)_i + \text{SUM}_i(Q_w)_i \\ \text{SUM}_i(R_g)_i &= \text{SUM}_i(dM_g/dt)_i + \text{SUM}_i(Q_g)_i \end{aligned}$$

## FULLY IMPLICIT AND IMPES METHODS

where  $SUM_i$  refers to the sum over all reservoir cells and  $(Ro)_i$  is the oil residual in cell  $i$  etc.. In ECLIPSE the material balance errors are converted to meaningful, problem independent, numbers by scaling to equivalent field saturation values;

$$\begin{aligned}MBo &= Boave.dt.SUM_i(Ro)_i/SUM_i(PV)_i \\MBW &= Bwave.dt.SUM_i(Rw)_i/SUM_i(PV)_i \\MBg &= Bgave.dt.SUM_i(Rg)_i/SUM_i(PV)_i\end{aligned}$$

where  $Boave$  is the average oil formation factor etc. The numerical values of  $MBo$ ,  $MBW$  and  $MBg$  are computed after each Newton iteration and the material balance errors are considered to be sufficiently small if they are all less than 0.0000001. For various reasons material balance errors are always very small in ECLIPSE and typical values of  $MB$  are 0.0000000000000001.  $MB$  values are printed out in the summary of each Newton iteration. Conventional material balance accounts in conventional units can also be printed at each report time.

### Normalised Residuals

---

Although great care is taken to ensure that material balance errors are unusually small in ECLIPSE, this is not considered to be a sufficiently rigorous test of convergence. The second test is obtained by computing the maximum saturation normalised residuals

$$\begin{aligned}CNVo &= Boave.dt.MAX_i |(Ro)_i/(PV)_i | \\CNVw &= Bwave.dt.MAX_i |(Rw)_i/(PV)_i | \\CNVg &= Bgave.dt.MAX_i |(Rg)_i/(PV)_i |\end{aligned}$$

where  $MAX_i$  is the maximum value over all cells in the reservoir. By converting each convergence error to an equivalent saturation value we can attach sensible limits to the  $CNV$  numbers, which are considered to have converged if they are all less than 0.001.

### Flows

---

The flow rate into cell  $i$  from a neighbouring cell  $n$ ,  $F_{ni}$ , is

FULLY IMPLICIT AND IMPES METHODS

$$F_{ni} = T_{ni} \begin{bmatrix} k_{ro}/(B_o V_o) & 0 & R_v.k_{rg}/(B_g V_g) \\ 0 & k_{rw}/(B_w V_w) & 0 \\ R_s.k_{ro}/(B_o V_o) & 0 & k_{rg}/(B_g V_g) \end{bmatrix} \begin{bmatrix} dP_{oni} \\ dP_{wni} \\ dP_{gni} \end{bmatrix}$$

where

$$\begin{aligned} dP_{oni} &= P_{on} - P_{oi} - DEN_{oni}.G.(D_n - D_i) \\ dP_{wni} &= P_{wn} - P_{wi} - DEN_{wni}.G.(D_n - D_i) \\ &= P_{on} - P_{oi} - DEN_{wni}.G.(D_n - D_i) - P_{cown} + P_{cowi} \\ dP_{gni} &= P_{gn} - P_{gi} - DEN_{gni}.G.(D_n - D_i) \\ &= P_{on} - P_{oi} - DEN_{gni}.G.(D_n - D_i) + P_{cogn} - P_{cogi} \end{aligned}$$

$T_{ni}$  is the transmissibility between cells n and i,  
 $k_r$  is the relative permeability (  $k_{ro}$  is the rel. perm. of oil etc. ),  
 $V$  is the viscosity (  $V_w$  is the viscosity of water etc. ),  
 $dP$  is the potential difference (  $dP_{gni}$  is the gas potential difference between cells n and i ),  
 $DEN$  is the fluid density (  $DEN_{oni}$  is the density of oil at the interface between cells n and i ),  
 $G$  is the acceleration due to gravity (.0000981 in metric units .00694 in field units and .000968 in lab units ),  
 $D$  is the cell centre depth.

The subscript u indicates that the fluid mobilities are to be evaluated in the upstream cell ( cell n if  $dP_{ni}$  is positive, cell i if  $dP_{ni}$  is negative ). The upstream calculation applies separately for each equation (oil,water,gas) so that, for example, oil may flow from cell i to cell n while water flows from cell n to cell i.

The net flow rate from cell i into neighbouring cells is obtained by summing over the neighbouring cells,  $F_i = \sum_n(F_{ni})$ .

The rate of flow into a production well from cell i is

$$Q_i = - T_{wi} * (P_{oi} - H_{iw} - P_{bh}) * \begin{bmatrix} k_{ro}/(B_o V_o) + R_v.k_{rg}/(B_g V_g) \\ k_{rw}/(B_w V_w) \\ k_{rg}/(B_g V_g) + R_s.k_{ro}/(B_o V_o) \end{bmatrix} \begin{bmatrix} P_{oi} - H_{iw} - P_{bh} \\ P_{oi} - H_{iw} - P_{bh} \\ P_{oi} - H_{iw} - P_{bh} \end{bmatrix}$$

where  $T_{wi}$  is the well connection transmissibility factor,  
 $H$  is the hydrostatic head correction and  
 $P_{bh}$  is the bottom hole pressure.

Well terms are discussed in detail in a separate appendix.

## FULLY IMPLICIT AND IMPES METHODS

### Reservoir Densities

---

Reservoir densities (DEN) of oil and gas are computed from surface densities (SD) using

$$\begin{aligned} \text{DENO} &= (\text{SDo} + \text{C.Rs.SDg})/\text{Bo} \\ \text{DENG} &= (\text{C.SDg} + \text{Rv.SDo})/\text{Bg} \end{aligned}$$

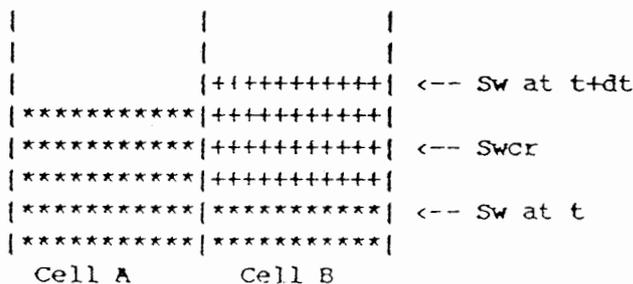
with  $C = 1$  for metric and lab units and  $C = 178.1076$  in field units. The reservoir density at the interface between cells  $n$  and  $i$  is computed as the average of the reservoir densities in cells  $n$  and  $i$

$$\begin{aligned} \text{DENoni} &= (\text{DENon} + \text{DENoi})/2 \\ \text{DENGni} &= (\text{DENGn} + \text{DENGi})/2 \end{aligned}$$

### Instability

---

The most common instability using IMPES arises when a phase crosses the critical saturation and becomes mobile. In the diagram below water is flowing from cell A to cell B. Water levels at time  $t$  are shown using asterisks (\*). Because the saturation in cell B is below  $\text{Swcr}$  at time  $t$ , water cannot flow out of cell B. During the timestep,  $dt$ , the water saturation in cell B can rise significantly above  $\text{Swcr}$  to a value greater than that in cell A. At the next timestep, water is allowed to flow out of cell B at a high rate especially if  $k_{rw}$  increases sharply above  $\text{Swcr}$ , and the saturation may fall below  $\text{Swcr}$  making water in cell B immobile again at the following timestep. Saturations may therefore oscillate about their critical values using IMPES. These difficulties do not arise with the fully-implicit method because as soon as  $\text{Sw}$  is greater than  $\text{Swcr}$  it is permitted to flow out of cell B.



This type of instability may limit the efficiency of the IMPES method. The problem is reduced in some simulators ( but not in ECLIPSE ) by automatically rescaling the input relative permeability curves to equally spaced saturation intervals. This is not done in ECLIPSE because:-

#### FULLY IMPLICIT AND IMPES METHODS

- (a) the user may wish his data to be honoured precisely,
- (b) sharp relative permeability contrasts are handled efficiently by the fully implicit method,
- (c) table look-up is equally efficient either way ( contrary to popular belief ), and
- (c) the user may input 'smooth' kr curves if he wishes.

## TRANSMISSIBILITY CALCULATIONS

---

### Transmissibility Calculations

---

This appendix describes the expressions used by ECLIPSE to calculate transmissibility values, between cells in the grid, and between cells and numerical aquifers.

There are three types of transmissibility calculation available in ECLIPSE. The first is the traditional block centred form, based upon the distances between the cell centres only. This is the default, and corresponds to the use of the OLDTRAN keyword in the GRID section. The second form also assumes a traditional block centred geometry but involves a different combination of cell cross-sectional areas and permeabilities from that assumed in the OLDTRAN formulation. The second form of transmissibility calculation is activated using the OLDTRANR keyword in the GRID section. The third form is based upon the use of cell corner points, which are available to ECLIPSE when the COORD/ZCORN form of grid definition is used. In this case it is possible to distinguish unambiguously between cell dip and fault displacement, and this option allows fault transmissibilities to be generated automatically. This third type of transmissibility calculation is specified by the use of the NEWTRAN keyword in the GRID section.

Although it is normal to use OLDTRAN with DX,DY,DZ,TOPS etc. input, and NEWTRAN with COORD/ZCORN, all types of transmissibility calculation are available with both types of grid specification, as ECLIPSE holds grid data internally in both forms. Corner points are obtained from DX/DY/DZ/TOPS data by assuming flat blocks, and DX/DY/DZ values are obtained from corner points by evaluation of the distances between cell face centre points. However, use of NEWTRAN with DX/DY/DZ/TOPS input may well produce large numbers of spurious fault connections, as the assumed flat grid blocks in neighbouring columns will generally overlap, and this combination of options is not recommended.

#### OLDTRAN Transmissibility Calculations.

---

In this case, x and y direction transmissibility values in the cartesian case are obtained using cell centre separations and cross sectional areas obtained from DX,DY and DZ, with a dip correction. The expressions for the cartesian case are given below :

X-transmissibility,

$$\text{TRANX}(I) = \text{CDARCY} \cdot \text{TMLTX}(I) \cdot \text{A} \cdot \text{DIPC}/\text{B}$$

TRANSMISSIBILITY CALCULATIONS

with :

TRANX(I) - The transmissibility between cell I and cell J, its neighbour in the positive X-direction  
 CDARCY - Darcy's constant ( .008527 in metric units, .001127 in field units and 3.6 in lab units )  
 TMLTX(I) - Transmissibility multiplier for cell I  
 A - Interface area between cell I and J  
 DIPC - Dip correction

A, DIPC and B are given by the expressions :

$$A = ( DX(J).DY(I).DZ(I).RNTG(I) + DX(I).DY(J).DZ(J).RNTG(J) ) / ( DX(I) + DX(J) )$$

$$B = ( DX(I)/PERMX(I) + DX(J)/PERMX(J) ) / 2$$

$$DIPC = DHS / ( DHS + DVS )$$

$$\text{with } DHS = ( (DX(I) + DX(J)) / 2 ) ** 2$$

$$DVS = ( DEPTH(I) - DEPTH(J) ) ** 2$$

RNTG is the net to gross ratio, which appears in the X- and Y-transmissibilities but not in the Z-transmissibility.

The expression for the Y-transmissibility value is entirely analogous to the above, with the appropriate permutations of X, Y and Z. The Z-transmissibility is given by :

$$TRANZ(I) = CDARCY.TMLTZ(I).A/B$$

with :

TRANZ(I) - The transmissibility between cell I and cell J, its neighbour in the positive Z-direction ( i.e. below cell I )  
 TMLTZ(I) - Z-transmissibility multiplier for cell I

Note that there is no dip correction in the Z-transmissibility case.

$$A = ( DZ(J).DX(I).DY(I) + DZ(I).DX(J).DY(J) ) / ( DZ(I) + DZ(J) )$$

$$B = ( DZ(I)/PERMZ(I) + DZ(J)/PERMZ(J) ) / 2$$

OLDTRANR Transmissibility Calculations.

---

In this case, x and y direction transmissibility values in the cartesian case are also calculated using cell centre separations and cross-sectional areas obtained from DX, DY and DZ, with a dip correction. The expressions for

TRANSMISSIBILITY CALCULATIONS

the cartesian case are given below :

X-transmissibility,

$$\text{TRANX}(I) = \text{CDARCY} \cdot \text{TMLTX}(I) \cdot \text{DIPC} / B$$

with :

- TRANX(I) - The transmissibility between cell I and cell J, its neighbour in the positive X-direction
- CDARCY - Darcy's constant in the appropriate units
- TMLTX(I) - Transmissibility multiplier for cell I
- DIPC - Dip correction

B and DIPC are given by the expressions :

$$B = 2.0 / ( \text{DX}(I) / (\text{PERMX}(I) \cdot \text{A}(I)) + \text{DX}(J) / (\text{PERMX}(J) \cdot \text{A}(J)) )$$

$$\begin{aligned} \text{with } \text{A}(I) &= \text{DY}(I) \cdot \text{DZ}(I) \cdot \text{RNTG}(I) \\ \text{A}(J) &= \text{DY}(J) \cdot \text{DZ}(J) \cdot \text{RNTG}(J) \end{aligned}$$

$$\text{DIPC} = \text{DHS} / (\text{DHS} + \text{DVS})$$

$$\begin{aligned} \text{with } \text{DHS} &= ( (\text{DX}(I) + \text{DX}(J)) / 2 )^{**2} \\ \text{DVS} &= ( \text{DEPTH}(I) - \text{DEPTH}(J) )^{**2} \end{aligned}$$

RNTG is the net to gross ratio, which appears in the X- and Y-transmissibilities but not in the Z-transmissibility.

The expression for the Y-transmissibility value is entirely analogous to the above, with the appropriate permutations of X, Y and Z. The Z-transmissibility is given by :

$$\text{TRANZ}(I) = \text{CDARCY} \cdot \text{TMLTZ}(I) / B$$

with :

- TRANZ(I) - The transmissibility between cell I and cell J, its neighbour in the positive Z-direction ( i.e. below cell I )
- TMLTZ(I) - Z-Transmissibility multiplier for cell I

Note that there is no dip correction in the Z-transmissibility case.

$$B = 2.0 / ( \text{DZ}(I) / (\text{PERMZ}(I) \cdot \text{A}(I)) + \text{DZ}(J) / (\text{PERMZ}(J) \cdot \text{A}(J)) )$$

$$\begin{aligned} \text{with } \text{A}(I) &= \text{DX}(I) \cdot \text{DY}(I) \\ \text{A}(J) &= \text{DX}(J) \cdot \text{DY}(J) \end{aligned}$$

## TRANSMISSIBILITY CALCULATIONS

### NEWTRAN Transmissibility Calculations.

---

In this case, the transmissibility values are calculated from the X-, Y- and Z-projections of the mutual interface area of the two cells. An inner product is then taken with the vector distance from the cell centre to the centre of the cell face, so that a dip correction is automatically incorporated. The X-transmissibility is given by the expression :

$$\text{TRANX}(I) = \text{CDARCY.TMLTX}(I)/(1.0/\text{TI}+1.0/\text{TJ})$$

$$\text{with TI} = \text{PERMX}(I).\text{RNTG}(I).(A.DI)/(DI.DI)$$

$$(A.DI) = \text{AX}.DIX+\text{AY}.DIY+\text{AZ}.DIZ$$

$$(DI.DI) = DIX**2+DIY**2+DIZ**2$$

AX,AY AND AZ are the X-,Y- and Z- projections of the mutual interface area of cell I and cell J ( which need not be neighbours in the cartesian indexing grid ) , and DIX, DIY and DIZ are the X-, Y- and Z-components of the distance between the centre of cell I and the centre of the relevant face of cell I, these centres being obtained as the appropriate average. The expression for TJ is analogous.

The Y- and Z- transmissibility expressions are similar, the net to gross ratio being absent from the Z expression.

### RADIAL Transmissibility Calculations.

---

There is no distinction between the method used to calculate radial transmissibility values whether grid data is supplied in DR/DTHETA/DZ/INRAD/TOPS form or COORD/ZCORN form. The expressions used are based upon true radial flow between the pressure equivalent radii, with a dip correction :

$$\text{TRANR}(I) = \text{CDARCY.TMLTR}(I).DIPC/(1.0/\text{TI}+1.0/\text{TJ})$$

$$\text{with TI} = \text{PERMR}(I).\text{RNTG}(I).DTHETA(I).DZ(I)/DIP$$

$$DIP = ( R1**2/(R2**2-R1**2).\text{LN}(R1/R2)+1/2 )$$

$$\text{and TJ} = \text{PERMR}(J).\text{RNTG}(J).DTHETA(J).DZ(J)/D2M$$

$$D2M = ( R3**2/(R3**2-R2**2).\text{LN}(R3/R2)-1/2 )$$

where R1 is the inner radius of cell I, R2 the outer radius of cell I and R3 the outer radius of cell J.

## TRANSMISSIBILITY CALCULATIONS

The azimuthal transmissibility is given by :

$$\text{TRANI}(I) = \text{CDARCY} \cdot \text{TMLTT}(I) / (1.0/\text{TI} + 1.0/\text{TJ})$$

$$\text{with TI} = 2 \cdot \text{PERMT}(I) \cdot \text{RNTG}(I) \cdot \text{DZ}(I) \cdot \text{LN}(R2/R1) / \text{DTHETA}(I)$$

where R1 is the inner radius, R2 the outer radius. TJ is similar.

LN( ) implies the natural logarithm throughout.

This expression is also used in completing the circle in radial geometry.

The vertical transmissibility is given by :

$$\text{TRANZ}(I) = \text{CDARCY} \cdot \text{TMLTZ}(I) / (1.0/\text{TI} + 1.0/\text{TJ})$$

$$\text{with TI} = \text{PERMZ}(I) \cdot \text{DTHETA}(I) \cdot (R2^{**2} - R1^{**2}) / \text{DZ}(I)$$

where R1 is the inner radius, R2 the outer radius. TJ is similar.

Note that the positions of the cells in a radial run may be entered by using COORD/ZCORN or INRAD/DR/DTHETA/DZ. In both cases the transmissibility expressions above may be used. There are some restrictions which exist on corner point positions for the above expressions to be sensible. Cells with the same J index should have a common DTHETA, and cells with the same I index should have a common DR. If COORD/ZCORN input is used, four corners of each cell should have a common inner radius, and four a common outer radius. Similarly, there should be a single upper and lower theta value. (This implies vertical coordinate lines in the radial case). The depths of cell corners are, however, arbitrary, and may be used to obtain dipping radial geometry.

### Transmissibility Calculations for Aquifers.

Aquifers are assumed to be one dimensional rows of cells, each with a cross-section, length and depth, but no further information concerning their shape. Transmissibility values are calculated using the OLDTRAN block centre method. The transmissibility between two aquifer cells is thus given by:

$$\text{TR} = \text{CDARCY} / (1.0/\text{TI} + 1.0/\text{TJ})$$

$$\text{with TI} = 2 \cdot \text{PERMX}(I) \cdot \text{XSECT}(I) / \text{LENGTH}(I)$$

where PERMX(I) is the permeability of aquifer cell I, XSECT(I) its cross-section and LENGTH(I) its length.

This expression is used for aquifer cell to aquifer cell transmissibilities in both radial and cartesian cases. However, in connecting the first aquifer cell to a grid block the radial or cartesian transmissibility from the block edge to centre is used.

## 5.0 DUAL POROSITY

This section provides an overview of the dual-porosity model and the assumptions made.

## DUAL POROSITY

---

### Dual Porosity

---

In a dual porosity reservoir, fluids exist in two interconnected systems: (a) the rock matrix, which usually provides the bulk of the reservoir volume and (b) the highly permeable rock fractures. If the matrix blocks are linked only through the fracture system, this conventionally could be regarded as a dual porosity single permeability system, since fluid flow through the reservoir takes place only in the fracture network with the matrix blocks acting as sources. If there is the possibility of flow directly between neighbouring matrix blocks, this is conventionally considered to be a dual porosity dual permeability system. Dual porosity dual permeability runs are computationally more expensive than dual porosity single permeability runs. A dual porosity single permeability run is specified by setting QDPORO = T and QDPERM = F in the RUNSPEC section.

To model such systems, two simulation cells are associated with each block in the geometric grid, representing the matrix and fracture volumes of the cell. In ECLIPSE, the porosity, permeability, depth etc. of these may be independently defined. A matrix-fracture coupling transmissibility is constructed automatically by ECLIPSE to simulate flow between the two systems due to fluid expansion, gravity drainage, capillary pressure etc.

In a dual porosity run of ECLIPSE the number of layers in the Z-direction should be doubled. ECLIPSE associates the first half of the grid ( the first NDIVIZ/2 layers ) with the matrix blocks, and the second half with the fractures. In such runs NDIVIZ must therefore be even; ECLIPSE will check that this is the case.

The intrinsic permeabilities of cells in the fracture system are equal to the specified values multiplied by the fracture porosities to yield effective fracture permeabilities. If the NODPPM keyword is used then the intrinsic permeability is used rather than the effective permeability.

If the dual porosity but not the dual permeability option is selected, the matrix blocks have no mutual transmissibilities. If dual porosity and dual permeability is chosen, the matrix blocks have their normal transmissibilities.

The matrix-fracture coupling transmissibility terms which exist between each cell of the matrix grid and the corresponding cell in the fracture grid are proportional to the cell bulk volume, being of the form :

$$TR = CDARCY.KPERM.VOL.SIGMA$$

where KPERM is taken as the X-direction permeability of the matrix blocks, VOL is the cell bulk volume ( note that this is not the pore volume, having

## DUAL POROSITY

no porosity factor ), and SIGMA is a factor of dimensionality  $LENGTH^{*-2}$ , to account for the matrix/fracture interface area per unit volume, i.e. the size of the blocks in the matrix volume. Kazemi (SPEJ Dec 76, 317-326) has proposed the following form for SIGMA :

$$SIGMA = 4 ( 1/lx^{*2} + 1/ly^{*2} + 1/lz^{*2} )$$

where lx, ly and lz are typical X, Y and Z dimensions of the blocks of material making up the matrix volume. ( lx, ly and lz are thus not related to the simulation grid dimensions). Alternatively, as SIGMA acts as a multiplier on the matrix-fracture coupling, it may simply be treated as a history matching parameter.

SIGMA is normally applied as a single parameter for the entire field. In practice, it may be necessary to vary SIGMA in different layers or from cell to cell. Although there is no facility to input SIGMA as a cell array in ECLIPSE, the desired effect can be achieved simply by making an appropriate adjustment to the X-direction permeabilities in the matrix blocks in a dual porosity run, because KPERM is used only to compute the matrix fracture coupling, TR.

An example of a dual porosity run is given in Sample Problem Data Set 5, and the use of the keywords required for dual porosity runs is described in the comments. In this case, the matrix and fracture blocks have identical depths: this might not be true in a run in which gravity drainage was to be considered.

The keyword DPGRID may be used to simplify the construction of grids for dual porosity runs. This keyword enables grid data to be entered for the matrix cells only ( the first NDIVIZ/2 layers ), the missing values for the remaining fracture layers being obtained from the corresponding matrix cell.

### Restrictions on dual porosity runs

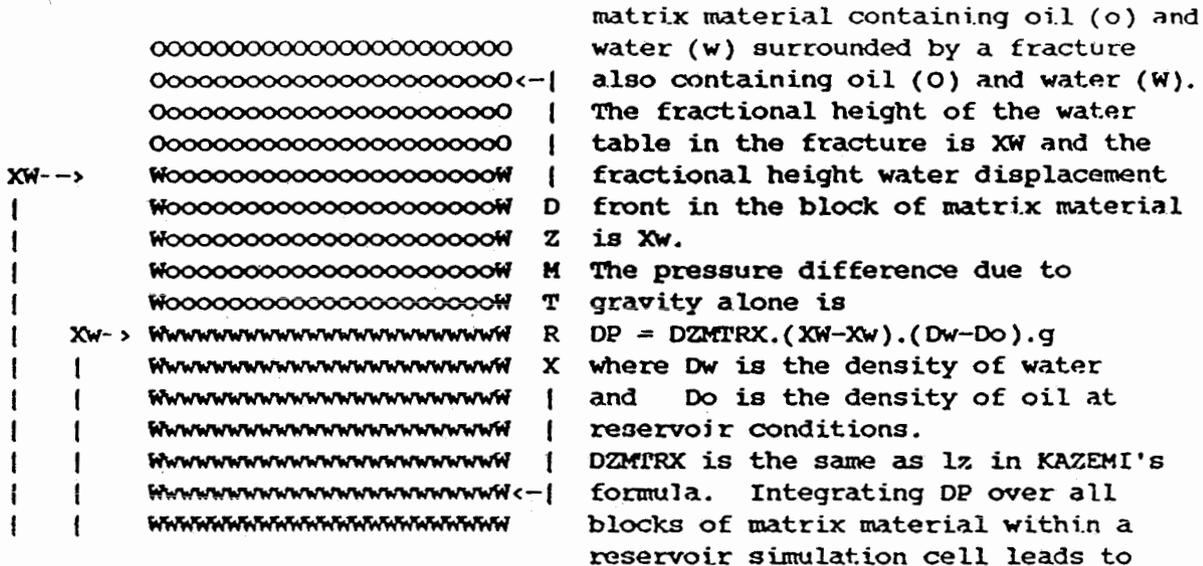
The following restrictions apply to dual porosity ( QDPORO = T QDPERM = F ) runs but NOT to dual porosity dual permeability ( QDPORO = T QDPERM = T ) runs.

- (a) Wells connect only to fracture cells - not to matrix cells.
- (b) Non-neighbour connections may not be used with matrix cells.  
The internal connection of each matrix cell to its appropriate fracture cell is made automatically by ECLIPSE.
- (c) Each active matrix cell must correspond with an active fracture cell.

DUAL POROSITY

Gravity Imbibition/Drainage

Fluid exchange between the fracture and matrix due to gravity is modelled by setting QGRAID = T in the RUNSPEC section and by giving a non-zero value to DZMTRX in the grid section. The illustration shows a typical block of



the same formula for DP except that it now represents the gravitational pressure difference between reservoir simulation fracture and matrix cells and XW and Xw are the partial volumes of mobile water in the simulation fracture and matrix cells. In ECLIPSE, the effect is modelled by introducing additional pseudo capillary pressures, PS

- PS<sub>cow</sub> = DZMTRX.XW.(Do-Dw).g - fracture cells
- PS<sub>cow</sub> = DZMTRX.Xw.(Do-Dw).g - matrix cells
- PS<sub>cog</sub> = DZMTRX.XG.(Do-Dg).g - fracture cells
- PS<sub>cog</sub> = DZMTRX.Xg.(Do-Dg).g - matrix cells
- PS<sub>cwg</sub> = DZMTRX.XG.(Dw-Dg).g - fracture cells
- PS<sub>cwg</sub> = DZMTRX.Xg.(Dw-Dg).g - matrix cells

Thus, the flow of free gas from a fracture to a matrix cell in a gas oil system is computed as

$$F_g = TR.GMOB.(P_{of}-P_{om}+dfm.D_g.g+P_{cogf}-P_{cogm}+DZMTRX.(X_G-X_g).(D_o-D_g)g )$$

where

- TR is the transmissibility between the fracture and matrix cells,
- GMOB is the gas mobility in the (upstream) fracture cell,
- P<sub>of</sub> is the oil phase pressure in the fracture cell,
- P<sub>om</sub> is the oil phase pressure in the matrix cell,
- dfm is the difference in depth between the fracture and matrix cells



## DUAL POROSITY

### Solution of the Linear Equations

---

The linear equations associated with dual porosity/permeability simulations can be written in the form

$$A.x + U.y = R_m$$

$$L.x + B.y = R_f$$

where A is the usual banded matrix for the matrix cells and B is the banded matrix for the fracture cells.  $R_m$  is the matrix residual and  $R_f$  is the fracture residual. L and U are diagonal matrices coupling the fracture and matrix systems. x and y are the solutions ( pressure and saturation changes ) in the matrix and fracture cells respectively.

In the dual porosity single permeability case, there is no direct communication between neighbouring matrix cells and the matrix A is diagonal. This greatly enhances the efficiency of the solution procedure used by ECLIPSE because the equations can now be simplified to the form

$$( B - L.AINV.U ) .y = R_f - L.AINV.R_m$$

$$x = AINV.( R_m - U.y )$$

where AINV is the inverse of A. Because L.AINV.U is diagonal it does not alter the banded structure of B. The problem size is effectively halved because the equation for y only involves fracture cells. The equations are solved sequentially, first for the fracture solution, y, and then for the matrix solution, x. In the more expensive dual porosity/dual permeability case above simplification does not apply and the equations are solved simultaneously.

DUAL POROSITY

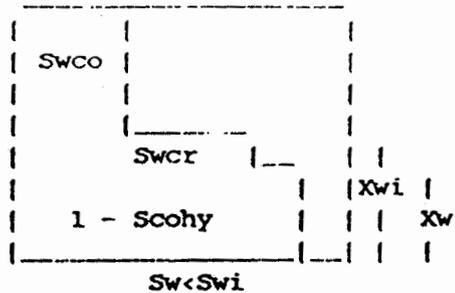
Calculation of Fractional Volumes

The initial fractional volume of water below the water contact,  $X_{wi}$ , is

$$X_{wi} = (S_{wi} - S_{wco}) / (1 - S_{cohy} - S_{wco})$$

where

- $S_{wi}$  is the initial water saturation
- $S_{wcr}$  is the critical water saturation
- $S_{wco}$  is the connate water saturation
- $S_{cohy}$  is the connate hydrocarbon saturation
- $S_{cohy} = S_{ocow}$  (oil/water systems)
- $= S_{ocow} + S_{gco}$  (oil/water/gas systems)
- $= S_{gco}$  (gas/water systems)
- $S_{scrhy}$  is the critical hydrocarbon saturation



If  $S_w > S_{wi}$  then

$$X_w = 1 \text{ when } S_w = S_{wmax} = X_{wi}(1 - S_{cohy}) + (1 - X_{wi})(1 - S_{scrhy})$$

$$X_w = (S_w - X_{wi}(S_{scrhy} - S_{cohy}) - S_{wco}) / (1 - S_{scrhy} - S_{wco}) \text{ when } S_w < S_{wmax}$$

If  $S_w < S_{wi}$  then

$$X_w = 0 \text{ when } S_w = S_{wmin} = X_{wi}.S_{wcr} + (1 - X_{wi}).S_{wco}$$

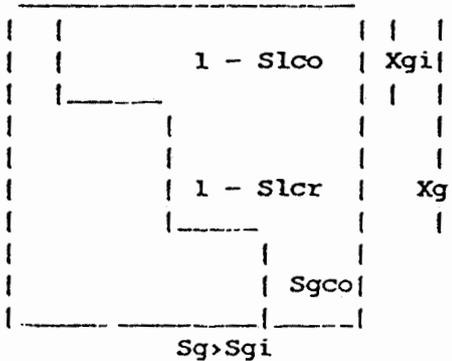
$$X_w = (S_w - X_{wi}(S_{wcr} - S_{wco}) - S_{wco}) / (1 - S_{cohy} - S_{wcr}) \text{ when } S_w > S_{wmin}$$

The initial fractional volume of gas above the gas contact,  $X_{gi}$ , is

$$X_{gi} = (S_{gi} - S_{gco}) / (1 - S_{lco} - S_{gco})$$

where

- $S_{gi}$  is the initial gas saturation
- $S_{gco}$  is the connate gas saturation
- $S_{gcr}$  is the critical gas saturation
- $S_{lco}$  is the connate liquid saturation
- $S_{lco} = S_{ocog}$  (oil/gas systems)
- $= S_{ocog} + S_{wco}$  (oil/water/gas systems)
- $= S_{wco}$  (gas/water systems)
- $S_{lcr}$  is the critical liquid saturation



If  $S_g > S_{gi}$  then

$$X_g = 1 \text{ when } S_g = S_{gmax} = X_{gi}(1 - S_{lco}) + (1 - X_{gi})(1 - S_{lcr})$$

$$X_g = (S_g - X_{gi}(S_{lcr} - S_{lco}) - S_{gco}) / (1 - S_{lcr} - S_{gco}) \text{ when } S_g < S_{gmax}$$

If  $S_g < S_{gi}$  then

$$X_g = 0 \text{ when } S_g = S_{gmin} = X_{gi}.S_{gcr} + (1 - X_{gi}).S_{gco}$$

$$X_g = (S_g - X_{gi}(S_{gcr} - S_{gco}) - S_{gco}) / (1 - S_{lco} - S_{gcr}) \text{ when } S_g > S_{gmin}$$

## 6.0 SATURATION FUNCTIONS

This section provides an overview of the three-dimensional relative permeability model. The discussions also include hysteresis in the relative permeability curves and the methodology for modeling. This section also includes the method for describing capillary equilibrium regions.

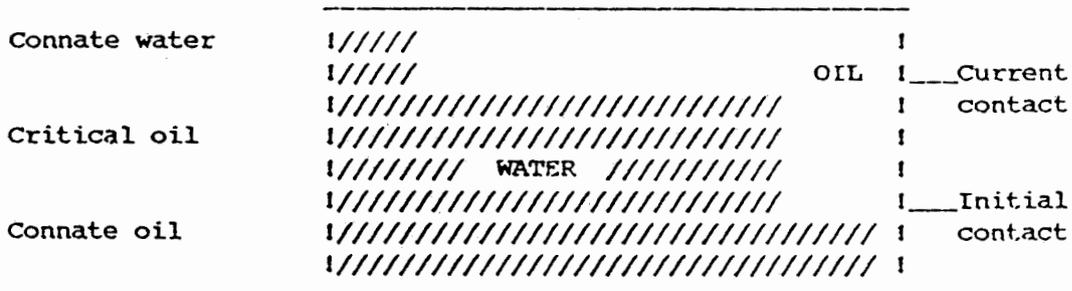
The ECLIPSE model includes a user specified vertical equilibrium options or instant gravity segregations on a block-by-block basis. This is also explained in this section.

VERTICAL EQUILIBRIUM

-----  
 Vertical Equilibrium  
 -----

This appendix describes the ECLIPSE vertical equilibrium option.

Vertical equilibrium (VE) is the assumption that equilibrium is established in a cell in a time short compared to that characteristic of horizontal flows. This implies that the phase hydrostatic potential is independent of depth within a cell. In the absence of capillary pressure effects, the saturation distribution is then a step function depending on the fluid contact depths. This saturation distribution is taken account of when ECLIPSE calculates relative permeability values for the cell faces. In the case shown, for example, water may flow out of the bottom of the cell, but not out of the top, at which no mobile water exists. The water flow through the sides reflects the fraction of the cell faces over which a mobile saturation exists. If the VE assumption of approximate hydrostatic equilibrium is valid, such a segregated saturation distribution may provide a more accurate description of fluid flow than a default dispersed option in which the fluids are assumed to be evenly distributed over the grid block.



The effect of corner point geometry upon the VE model is to introduce different depths for cell faces. For example, water may only flow out of an updip cell face if the contact level is sufficiently high.

In such a model hysteresis can occur if, for example, oil invades the water zone and retreats, leaving a residual oil saturation.

An option exists in the ECLIPSE VE implementation which enables the user to specify the fractions of distributed and segregated saturation distribution functions which are to be used.

The use and formulation of the ECLIPSE VE option is discussed in more detail below.

## VERTICAL EQUILIBRIUM

### Using the ECLIPSE VE Option

---

The vertical equilibrium option in ECLIPSE is enabled by setting the QVEOPT flag in the RUNSPEC section of the input data. One keyword, VEFAC, in the PROPS section, is specific to VE. This takes one real argument, which must lie in the range 0.0 to 1.0. This enables the user to specify dispersed ( rock curve ) and segregated ( VE ) saturation fractions. For example, using the following :

```
VEFRAC  
0.8 /
```

will specify 80% VE and 20% rock curves. The default is a VEFAC value of 1.0. If the VE option is not required, set QVEOPT to the default value of F, rather than using a VEFAC value of 0.0.

If the QVEOPT flag is set, and the ninth argument of the EQUIL keyword data is set to zero, then a method of equilibration specific to VE is used, which ensures that the initial contact depths are exactly matched by the initial saturations, and which yields a static initial solution. If this ninth argument is not zero, then a normal ECLIPSE fine grid equilibration is performed using the rock curves.

The latter method is only likely to be useful if a non default VEFAC value is used ; for a pure VE run, the former option is best.

To obtain a printed output of the contact depths at each report step, the 22nd and 23rd arguments of the RPTSCHED keyword may be set to 1. These control the output of oil water and oil gas contact depths respectively.

VERTICAL EQUILIBRIUM

The Cell Saturation Distribution

---

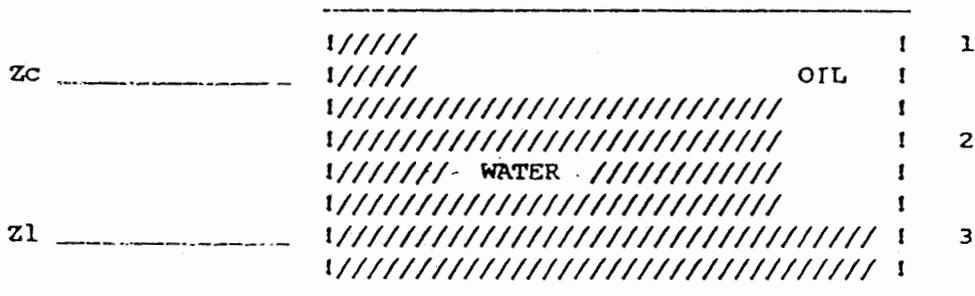
The saturation distribution in a cell assumed by the VE model is based on the assumption that the phase hydrostatic potential is independent of depth. In the absence of capillary pressure effects, the saturation distribution becomes a step function. The form of this depends on the phase option used.

1. The oil water case.

The distribution is derived from the water saturation value,  $S_w$ . If  $Z_c$  is the current oil water contact depth, and  $Z_l$  is the lowest contact depth which has occurred in the run, the distribution is as shown below. This has three depth intervals, as follows:

- Region 1 : Above the current oil water contact. The water saturation takes the connate value,  $S_{wco}$ , and the oil saturation is  $1-S_{wco}$ .
- Region 2 : Above the current oil water contact, but below the current. In this region water has displaced oil, so the critical oil saturation remains,  $S_{ocr}$ , whilst the water saturation is  $1-S_{ocr}$ .
- Region 3 : Below the lowest oil water contact. Only the connate oil saturation exists, which is usually zero, and the water saturation is  $1-S_{oco}$ .

If the current contact attempts to fall below  $Z_l$ , oil invading the water zone, then  $Z_l$  is set to the new  $Z_c$  value. If the contact again rises  $Z_l$  retains its value, so that a critical oil saturation remains in the invaded region.



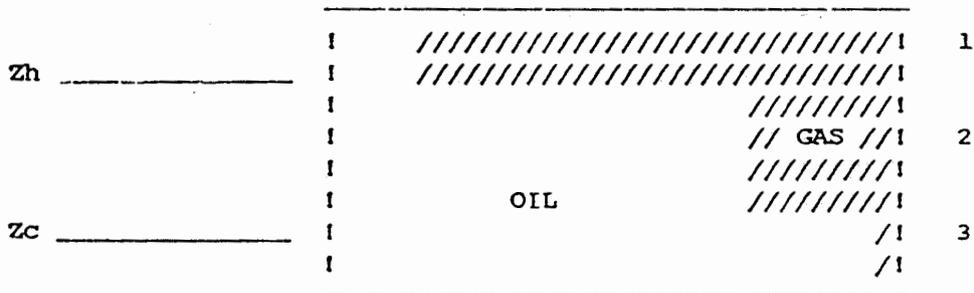
VERTICAL EQUILIBRIUM

2. The oil gas case.

The distribution is derived from the gas saturation value,  $S_g$ . If  $Z_c$  is the current oil gas contact depth, and  $Z_h$  is the highest contact depth which has occurred in the run, the distribution is as shown below. This has three depth intervals, as follows:

- Region 1 : Above the highest oil gas contact. The connate oil saturation exists,  $S_{oco}$ , which is often zero, and the gas saturation is  $1-S_{oco}$ .
- Region 2 : Below the highest oil gas contact, but above the current. In this region gas has displaced oil, so the critical oil saturation remains,  $S_{ocr}$ , whilst the gas saturation is  $1-S_{ocr}$ .
- Region 3 : Below the current oil gas contact. The gas saturation is the connate value,  $S_{gco}$ , which is usually zero, and the oil saturation is  $1-S_{gco}$ .

If the current contact attempts to rise above  $Z_h$ , oil invading the gas zone, then  $Z_h$  is set to the new  $Z_c$  value. If the contact again falls  $Z_h$  retains its value, so that a critical oil saturation remains in the invaded region.



VERTICAL EQUILIBRIUM

3. The gas water case.

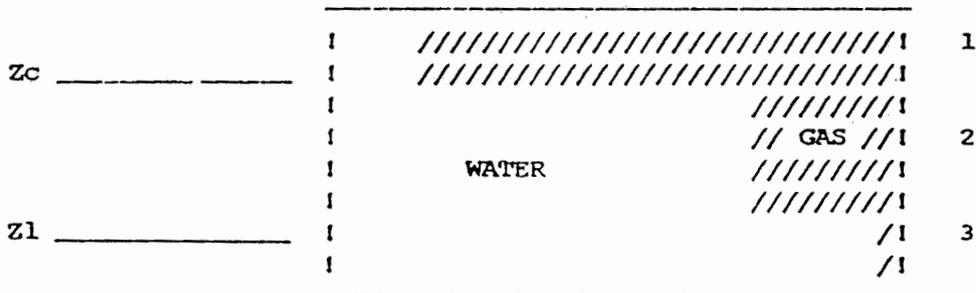
The distribution is derived from the gas saturation value,  $S_w$ . If  $Z_c$  is the current water gas contact depth, and  $Z_l$  is the lowest contact depth which has occurred in the run, the distribution is as shown below. This also has three depth intervals, as follows:

Region 1 : Above the current water gas contact. The water saturation has the connate value,  $S_{wco}$ , and the gas saturation is  $1 - S_{wco}$ .

Region 2 : Above the lowest water gas contact, but above the current. In this region water has displaced gas, so the critical gas saturation remains,  $S_{gcr}$ , whilst the water saturation is  $1 - S_{wcr}$ .

Region 3 : Below the lowest water gas contact. The gas saturation is the connate value,  $S_{gco}$ , which is usually zero, and the water saturation is  $1 - S_{gco}$ .

If the current contact attempts to fall below  $Z_l$ , gas invading the water zone, then  $Z_l$  is set to the new  $Z_c$  value. If the contact again falls  $Z_h$  retains its value, so that a critical water saturation remains in the invaded region.



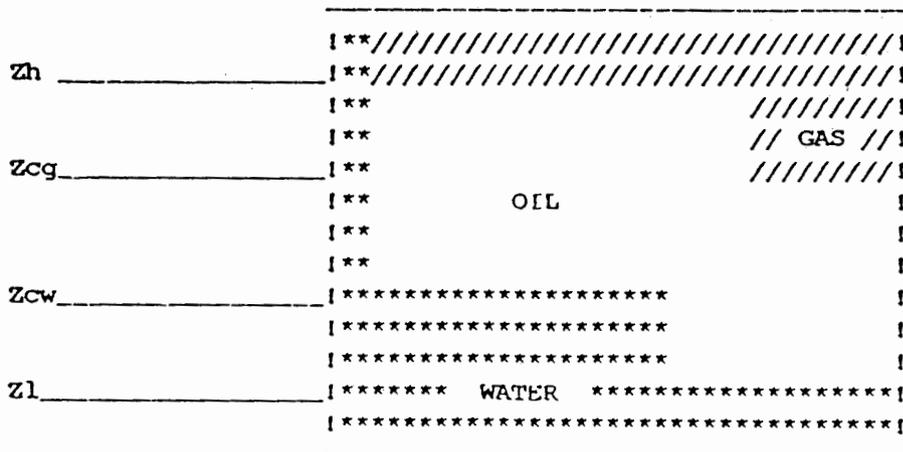
VERTICAL EQUILIBRIUM

4. The three phase case.

In this case  $S_w$  and  $S_g$  are distinct variables. The usual distribution is shown in the figure below. There are five depth intervals, as follows :

- Region 1 : Above the highest gas oil contact. The water saturation has the connate value,  $S_{wco}$ , and the oil saturation is the connate with respect to gas,  $S_{ocog}$ . The gas saturation is thus  $1-S_{wco}-S_{ocog}$ .
- Region 2 : Below the highest oil gas contact, but above the current. Gas has displaced oil, so the critical oil saturation with respect to gas remains,  $S_{ocrg}$ , with  $S_w=S_{wco}$  and  $S_g=1-S_{ocrg}-S_{wco}$ .
- Region 3 : Below the current oil gas contact and above the current oil water contact. The water and gas saturations take their connate values,  $(S_{wco}, S_{gco})$ , and the oil saturation is  $1-S_{wco}-S_{gco}$ .
- Region 4 : Below the current oil water contact, but above the lowest oil water contact. Water has displaced oil, so the critical oil saturation with respect to water remains,  $S_{ocrw}$ , with  $S_w=1-S_{ocrw}-S_{gco}$ .
- Region 5 : Below the lowest oil water contact. The gas saturation is connate,  $S_{gco}$ , and the oil saturation is the connate value with respect to water,  $S_{ocow}$ . The water saturation is  $1-S_{ocow}-S_{gco}$ .

In any of the phase combinations, one or more of the regions may not exist, depending on the conditions in a cell.

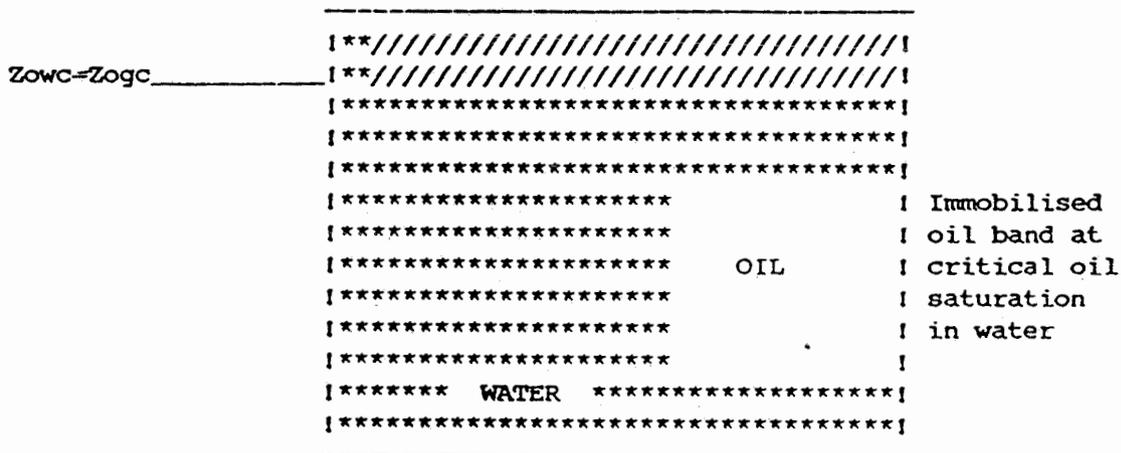


In the 85/7 version of Eclipse the oil-water contact was simply not permitted to be above the oil-gas contact. In the current 86/1 version the treatment of the three-phase VE case has been extended to cover

VERTICAL EQUILIBRIUM

the case in which the oil zone becomes immobilised. When this occurs, the oil zone becomes a residual saturation over a certain depth interval, and the water may rise above this zone, leading to a gas-water contact.

The immobilised oil state can occur with the gas-water contact above, below or in the immobilised oil. The first situation would arise if the water contact rose up into the original oil zone. In the normal three phase oil case, the residual oil saturation would then have to exist from the original contact depth to the current. This is possible as long as sufficient oil saturation exists. If, however, the contact continues to rise a point will be reached at which all the oil exists at residual saturation. If the contact rises further, the following immobilised oil distribution will exist ( assuming connate oil and gas saturations are zero ) :



ECLIPSE now models this immobilised oil state correctly, and will return reversably to the mobile oil state once a sufficient oil saturation exists.

## SATURATION FUNCTIONS

---

### Saturation Functions

---

There are four keywords that define the saturation-dependent properties of the reservoir fluids:

- SWFN - Sets water relative permeability and capillary pressure as a function of water saturation. This is required for 2 or 3 phase systems with water.
- SGFN - Sets gas relative permeability and capillary pressure as a function of gas saturation. This is required for 2 or 3 phase systems with gas.
- SOF3 - Sets relative permeability of oil in water, and oil in gas at the connate water saturation, as a function of oil saturation. This is required for 3 phase systems.
- SOF2 - Sets oil relative permeability as a function of oil saturation. This is required for 2 phase systems with oil.

### Water Saturation Properties

---

An example data table is given below:

—	SWAT	KRW	PCOW
—			(PSIA)
SWFN			
	.22	.0	7.0
	.3	.07	4.0
	.4	.15	3.0
	.5	.24	2.5
	.6	.33	2.0
	.8	.65	1.0
	.9	.83	.5
	1.0	1.0	.0 /

The water saturations must be entered in ascending order.

The third column is the oil-water capillary pressure ( $P_{cow} = P_o - P_w$ ). The values can become negative with increasing water saturation, if required. They should all be set to zero if there is no water-oil capillary pressure. In saturation tables intended for well completions only (i.e. not referenced for any grid block in keyword SATNUM) the capillary pressure data is not used, and the values entered are disregarded. In gas-water problems, the water

## SATURATION FUNCTIONS

capillary pressure is interpreted as the gas-water capillary pressure ( $P_{cgw} = P_g - P_w$ ).

Three saturation values in the table are of special interest:

- Swcr** - The CRITICAL water saturation.  
This is the highest saturation value for which the relative permeability is zero. At saturations above this value, water is mobile. Note that a critical water saturation MUST be defined ( the SWFN table must contain a water saturation for which  $k_{rw}=0$  ).  
In the example above,  $Swcr = 0.22$
- Swco** - The minimum water saturation value in the table.  
The Equilibration calculation (see keyword EQUIL) sets the water saturation to this value in grid blocks that lie above the water contact (or water transition zone). In this respect Swco is the CONNATE water saturation.  
In the example shown above,  $Swco = Swcr$  . If for any reason the connate water saturation is less than the critical value, the table should begin with the connate value. E.g. for  $Swco = 0.2$  and  $Swcr = 0.22$  , the table should begin with the lines
- |      |     |     |
|------|-----|-----|
| .20  | .0  | 7.0 |
| .22  | .0  | 7.0 |
| .3   | .07 | 4.0 |
| etc. |     |     |
- Swmax** - The maximum water saturation value in the table.  
The Equilibration calculation sets the water saturation to this value in grid blocks that lie below the water transition zone.  
In the example shown above,  $Swmax = 1.0$  , which results in the water zone being fully saturated with water.

### Gas Saturation Properties

An example data table is given below:

## SATURATION FUNCTIONS

-- SGAS	KRG	PCOG (PSIA)
SGFN		
.0	.0	0.0
.04	.0	.2
.1	.022	.5
.2	.1	1.0
.3	.24	1.5
.4	.34	2.0
.5	.42	2.5
.6	.5	3.0
.7	.81	3.5
.78	1.0	3.9 /

The gas saturations must be entered in ascending order.

The third column is the oil-gas capillary pressure ( $PCOG = P_g - P_o$ ). The values can start negative if required. They should all be set to zero if there is no gas-oil capillary pressure. In saturation tables intended for well completions only (i.e. not referenced for any grid block in keyword SATNUM) the capillary pressure data is not used, and the values entered are disregarded. In gas-water problems, the gas-water capillary pressure is defined in the water saturation table, so the capillary pressure values in the gas saturation table should be set to zero.

Three saturation values in the table are of special interest:

- Sgcr - The CRITICAL gas saturation.  
This is the highest saturation value for which the relative permeability is zero. At saturations above this value, gas is mobile. Note that a critical gas saturation MUST be defined ( the SGFN table must contain a gas saturation for which krg=0 ).  
In the example above, Sgcr = 0.04 .
  
- Sgco - The minimum gas saturation value in the table.  
The Equilibration calculation (see keyword EQUIL) sets the gas saturation to this value in grid blocks that lie below the gas contact (or water transition zone). In this respect Sgco is the CONNATE gas saturation.  
Normally Sgco = 0.0 , as in the example above.
  
- Sgmax - The maximum gas saturation value in the table.  
The Equilibration calculation sets the gas saturation to this value in grid blocks that lie above the gas transition zone.  
Normally Sgmax = 1.0 - Swco , as in the example above.

## SATURATION FUNCTIONS

### Oil Saturation Properties

---

The SOF2 and SOF3 tables are used in the calculation of oil relative permeabilities. They are not used in the calculation of the initial saturations. The ECLIPSE equilibration calculation sets oil saturations using the formula

$$S_o = 1 - S_w - S_g$$

An example data table for a 3 phase system is given below:

SOIL	KROW	KROG
.0000	.0000	.0000
.2000	.0000	.0000
.3800	1*	.0000
.4000	.0050	1*
.4800	1*	.0200
.5000	.0649	1*
.5800	1*	.1000
.6000	.1250	1*
.6800	1*	.3300
.7000	.4000	1*
.7400	1*	.6000
.7800	1.0000	1.0000

The oil saturations must be entered in ascending order in column 1. Columns 2 and 3 contain the corresponding oil relative permeabilities for oil-water systems, and for oil-gas-connate water systems respectively. Columns 2 and 3 may contain default specifications ( represented by 1\* ). When the table is read in, defaults are replaced by values obtained by linear interpolation ( in the above example, the first default in column 2 is replaced by the value 0.0045 )

Two saturation values in the table are of special interest:

- Socr - The CRITICAL oil saturation.  
This is the highest saturation value for which BOTH the oil-water and oil-gas permeabilities are zero. At saturations above this value, oil is mobile. Note that a critical oil saturation MUST be defined ( the SOF3 table must contain an oil saturation for which both krow and krog are 0 ).  
In the example above, Socr = 0.20 .
- Somax - The maximum oil saturation value in the table.  
This should be equal to 1 - Swco as determined from the SWFN

## SATURATION FUNCTIONS

table. The two oil relative permeabilities should be the same at  $S_{omax}$  ( both represent a case with  $S_o=S_{omax}$ ,  $S_w=S_{wco}$  and  $S_g=0$  ). ECLIPSE reports an error if this condition is not satisfied.

### Three Phase Oil Relative Permeability Models

---

A choice of three different formulae are available in ECLIPSE to allow users to determine the sensitivity of a reservoir model to a particular form of the three phase oil relative permeability. The default model for the three phase oil relative permeability is based on an assumption of complete segregation of the water and gas within each grid cell. The model provides a simple but effective formula which avoids the problems associated with other methods ( poor conditioning, negative values etc. ). ECLIPSE also contains options for using either of the modified three phase oil relative permeability models suggested by Stone. These models are summarised in 'Petroleum Reservoir Simulation', Aziz and Settari, p33 .

The default model assumed by ECLIPSE is shown in Figure 1. The oil saturation is assumed to be constant and equal to the block average value,  $S_o$ , throughout the cell. The gas and water are assumed to be completely segregated, except that the water saturation in the gas zone is equal to the connate saturation,  $S_{wco}$ . The full breakdown, assuming the block average saturations are  $S_o$ ,  $S_w$  and  $S_g$  ( with  $S_o + S_w + S_g = 1$  ) is as follows:

In a fraction  $S_g/(S_g+S_w-S_{wco})$  of the cell (the gas zone),

the oil saturation is  $S_o$   
the water saturation is  $S_{wco}$   
the gas saturation is  $S_g + S_w - S_{wco}$

In a fraction  $(S_w-S_{wco})/(S_g+S_w-S_{wco})$  of the cell (the water zone),

the oil saturation is  $S_o$   
the water saturation is  $S_g + S_w$   
the gas saturation is 0

The oil relative permeability is then given by

$$k_{ro} = \frac{( S_g.k_{rog} + (S_w-S_{wco}).k_{row} )}{( S_g + S_w - S_{wco} )}$$

where  $k_{rog}$  is the oil relative permeability for a system with oil, gas and connate water ( tabulated as a function of  $S_o$  ), and  $k_{row}$  is the oil relative permeability for a system with oil and water only ( also tabulated as a function of  $S_o$  ).

SATURATION FUNCTIONS

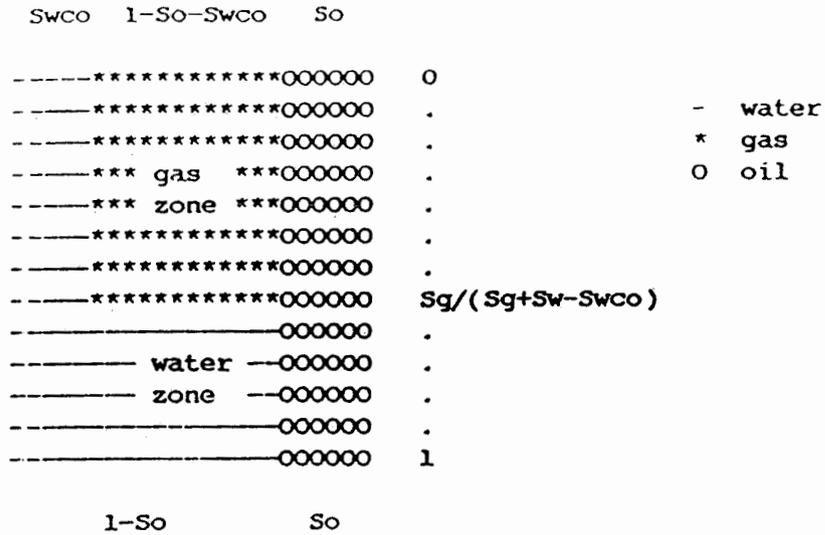


Figure 1

The second model available in ECLIPSE for calculating values of the three phase oil relative permeability is a modified version of the first model suggested by Stone ( Trans AIME,249,1970,214-218 ). The formula is

$$k_{ro} = k_{rocw} \cdot S_{So} \cdot F_w \cdot F_g$$

where

$k_{rocw}$  = value of the oil relative permeability in the presence of connate water only

$$S_{So} = ( S_o - S_{om} ) / ( 1 - S_{wco} - S_{om} ) \quad \text{when } S_o > S_{om}$$

$$F_w = k_{row} / ( k_{rocw} \cdot ( 1 - S_{Sw} ) )$$

$$F_g = k_{rog} / ( k_{rocw} \cdot ( 1 - S_{Sg} ) )$$

where

$$S_{Sw} = ( S_w - S_{wco} ) / ( 1 - S_{wco} - S_{om} ) \quad \text{when } S_w > S_{wco}$$

$$S_{Sg} = S_g / ( 1 - S_{wco} - S_{om} )$$

## SATURATION FUNCTIONS

In these formulae  $S_o$ ,  $S_w$  and  $S_g$  denote block averaged values for the oil, water and gas saturations in a grid cell.  $k_{rog}$  denotes the oil relative permeability for a system with oil, gas and connate water, and  $k_{row}$  denotes the oil relative permeability for a system with oil and water only. Both two phase oil relative permeability functions are tabulated as functions of oil saturation in the input data.  $S_{om}$  is the minimum residual oil saturation.  $k_{rocw}$  denotes the oil relative permeability in the presence of connate water only.

The calculation of three phase oil relative permeability using Stone's method 1 is enabled by means of `STONE1` keyword in the `PROPS` section of the input data. Tabulated output of the three phase oil relative permeability values are controlled by the 1st. argument of the `RPTPROPS` keyword.

The third model provided in ECLIPSE for calculating values of the three phase oil relative permeability is a modified form of the second model suggested by Stone ( J. Can. Pet. Tech., 12, 1973, 53-61 ). The formula is

$$k_{ro} = k_{rocw} \cdot ( (k_{row}/k_{rocw} + k_{rw}) \cdot (k_{rog}/k_{rocw} + k_{rg}) - k_{rw} - k_{rg} )$$

where  $k_{rog}$  denotes the oil relative permeability for a system with oil, gas and connate water;  $k_{row}$  denotes the oil relative permeability for a system with oil and water only. Both sets of two phase oil relative permeability functions are tabulated as functions of oil saturation in the input data.  $k_{rocw}$  is the oil relative permeability in the presence of connate water only.

Note the values of  $k_{ro}$  produced by this formula can be negative. The problem can be overcome by setting a lower bound of zero on the values of  $k_{ro}$ .

The calculation of three phase oil relative permeability using Stone's method 1 is enabled by means of `STONE2` keyword in the `PROPS` section of the input data. Tabulated output of the three phase oil relative permeability values are controlled by the 1st. argument of the `RPTPROPS` keyword.

The two Stone's models should not be used at present with either the hysteresis or the saturation table end point scaling options.

### Table End Points

In the Equilibration calculation (see keyword `EQUIL`) the water and gas saturation values above and below the transition zones are determined from the maximum and minimum saturation values in the water and gas saturation tables. The oil saturation values are thus defined consequentially from these values. The initial saturations in each zone are shown in Figure 2 .

SATURATION FUNCTIONS

---

GAS ZONE

$$S_g = S_{gmax}$$

$$S_w = S_{wco}$$

$$S_o = 1 - S_{gmax} - S_{wco}$$

---

GAS - OIL TRANSITION ZONE

---

OIL ZONE

$$S_g = S_{gco}$$

$$S_w = S_{wco}$$

$$S_o = 1 - S_{gco} - S_{wco}$$

---

OIL - WATER TRANSITION ZONE

---

WATER ZONE

$$S_g = S_{gco}$$

$$S_w = S_{wmax}$$

$$S_o = 1 - S_{gco} - S_{wmax}$$

---

Figure 2

---

## SATURATION FUNCTIONS

### Consistency Requirements

---

The oil, water and gas saturation tables, for each particular saturation table number region, must obey certain consistency requirements. These are detailed below.

1.  $S_{gmax}$  must not exceed  $1 - S_{wco}$

If this condition is violated, the gas saturation in the gas cap will be re-set to  $1 - S_{wco}$ , to prevent a negative oil saturation. Normally, if there is no oil in the gas cap,  $S_{gmax} = 1 - S_{wco}$ .

2.  $S_{gco}$  must not exceed  $1 - S_{wmax}$

If this condition is violated, the gas saturation in the water zone will be re-set to  $1 - S_{wmax}$ , to prevent a negative oil saturation. Normally, there is no initial free gas below the gas cap and the water zone is fully saturated with water, thus  $S_{gco} = 0$  and  $S_{wmax} = 1$ .

3.  $S_{omax}$  must equal  $1 - S_{wco}$

4.  $k_{row}(S_{omax})$  must equal  $k_{rog}(S_{omax})$

5.  $k_{rw}(S_w=0) = k_{rg}(S_g=0) = k_{row}(S_o=0) = k_{rog}(S_o=0) = 0$ .

Otherwise, phases can be mobile even at zero saturation and there is nothing to stop saturations going negative.

## HYSTERESIS

---

### Hysteresis

---

This appendix describes the ECLIPSE hysteresis option.

This option enables the user to specify different saturation functions for drainage ( decreasing wetting phase saturation ) and imbibition ( increasing wetting phase saturation ) processes.

The user supplies two saturation function table numbers for each cell, using the SATNUM and IMBNUM keywords in the REGIONS section. These provide respectively the primary drainage and pendular imbibition curves.

The primary drainage curve is for a process which starts at the maximum possible wetting phase saturation,  $S_{wmaxd}$ . ( This value will depend upon the endpoints of the saturation tables specified using the SATNUM keyword.) If the wetting phase saturation decreases to  $S_{wmin}$ , this primary drainage curve is used.

In a similar way, if the initial saturation is  $S_{wmin}$ , and the wetting phase saturation increases to  $S_{wmaxi}$ , the imbibition table data will be used. ( The maximum wetting phase saturation which can be reached,  $S_{wmaxi}$ , is determined from the endpoints of the tables specified using the IMBNUM keyword, and will generally be less than  $S_{wmaxd}$ . )

If the drainage or imbibition process is reversed at some point, the data used does not simply run back over its previous values. The way in which this hysteresis process is treated in ECLIPSE follows the techniques described in 'Petroleum Reservoir Simulation', Aziz and Settari, p398 ; J.E.Killough, Trans. AIME, 261, 1976, p37 ; and F.M Carlson, SPE 10157, San Antonio, 1981.

## HYSTERESIS

### Using the ECLIPSE Hysteresis Option

---

The hysteresis option in ECLIPSE is enabled by setting the QHYSTR option to T in the RUNSPEC section of the input data. The normal saturation table numbers in the REGIONS section are taken as defining the drainage curves. A new set of keywords, IMBNUM, IMBNUMX, IMBNUMX-, ..., IMBNUMZ-, are used to specify the tables to be used for each cell to supply imbibition curves. If the imbibition and drainage table numbers for a cell are the same, no hysteresis will occur for that cell.

The 11th to the 17th arguments of the RPTREGS keyword will control the output of the entered imbibition table region number data.

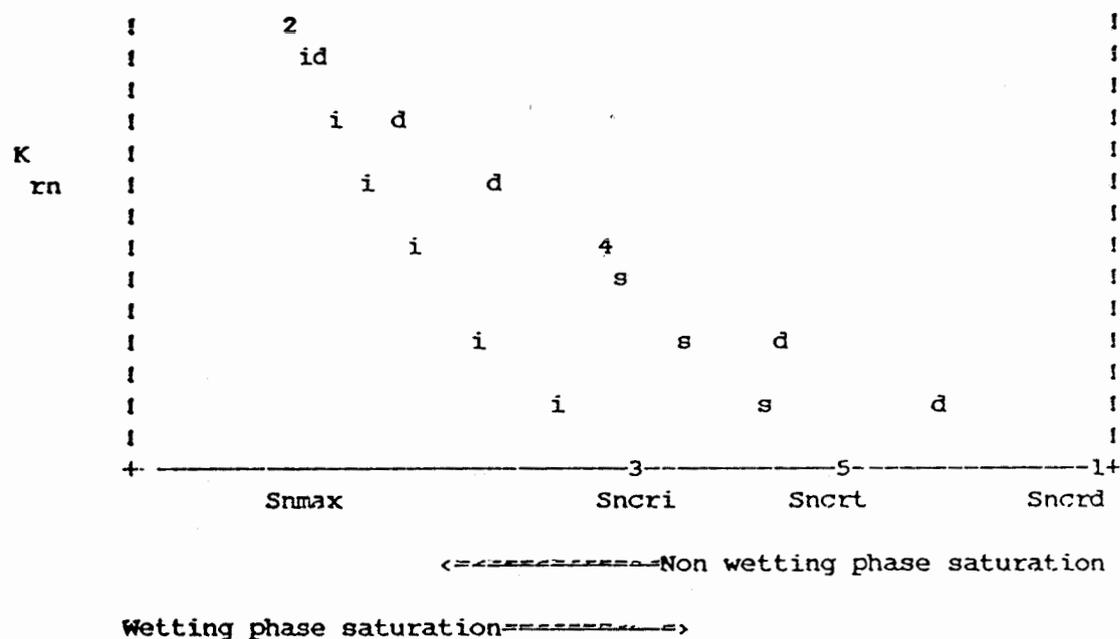
A further keyword, EHYSTR in the PROPS section, is specific to hysteresis. This sets the values of two parameters that determine the form of the scanning curves for capillary pressure hysteresis and wetting phase relative permeability hysteresis, and selects one of a choice of models for relative permeability hysteresis.

The reservoir is always equilibrated using the drainage curves.

## HYSTERESIS

### Relative Permeability Hysteresis in the Non-Wetting Phase

The non-wetting phases are oil (in an oil-water or three-phase run) and gas (in an oil-gas, water-gas or three-phase run). A typical pair of relative permeability curves for a non-wetting phase is shown below. The curve marked with d's represents the user-supplied drainage relative permeability table, and the curve marked with i's represents the user-supplied imbibition relative permeability table. (Note that non-wetting phase saturation increases from right to left in this diagram). The critical saturation of the imbibition curve ( $S_{ncri}$ ) is greater than that of the drainage curve ( $S_{ncrd}$ ). The two curves **MUST** meet at the maximum saturation value ( $S_{nmax}$ ).



Consider a drainage process starting at point 1. If a full drainage process is carried out, the bounding drainage curve is followed to point 2. If an imbibition process then occurs, the water saturation increasing, the bounding imbibition curve is followed to point 3, the imbibition critical saturation.

But suppose that the drainage process is reversed at some intermediate point 4. A scanning curve results, which is marked by s's in the diagram. The critical saturation remaining at point 5 is the trapped critical saturation ( $S_{ncrt}$ ), which is a function of the maximum non-wetting phase saturation reached in the run ( $S_{ny}$ ).

If a further drainage process begins from any point on the scanning curve 5 to 4, the same scanning curve is retraced until  $S_{ny}$  is reached, at which

## HYSTERESIS

point the drainage curve is rejoined.  $S_{hy}$  is updated during the run, so that further imbibition processes would occur along the appropriate scanning curves.

There is a choice of two methods for the generation of scanning curves from a given value of  $S_{hy}$ : Carlson's method and Killough's method. These methods are described in the papers referenced above. The choice of method is governed by Item 2 in keyword EHYSTR.

Carlson's method produces a scanning curve that is parallel to the imbibition curve. It can be visualised by shifting the imbibition curve horizontally until it cuts the drainage curve at the saturation  $S_{hy}$ . When this method is chosen, it is important to ensure that the imbibition curve is always steeper than the drainage curve at the same  $K_r$  value. If this is not the case, the scanning curve could cross to the right of the drainage curve, which may produce a negative value of  $S_{ncrt}$ .

Killough's method does not have such a simple geometric interpretation. For a given value of  $S_{hy}$ , the trapped critical saturation is calculated as

$$S_{ncrt} = S_{ncrd} + ( S_{hy} - S_{ncrd} ) / [ 1 + C ( S_{hy} - S_{ncrd} ) ]$$

where

$$C = 1 / ( S_{ncri} - S_{ncrd} ) - 1 / ( S_{nmax} - S_{ncrd} )$$

(Killough's formulae have been adapted to allow for non-zero values of  $S_{ncrd}$ .) The relative permeability for a particular saturation  $S_n$  on the scanning curve is

$$K_{rn}(S_n) = K_{rni}(S_{norm}) K_{rnd}(S_{hy}) / K_{rnd}(S_{nmax})$$

where  $K_{rni}$  and  $K_{rnd}$  represent the relative permeability values on the bounding imbibition and drainage curves respectively, and

$$S_{norm} = S_{ncri} + ( S_n - S_{ncrt} ) ( S_{nmax} - S_{ncri} ) / ( S_{hy} - S_{ncrt} )$$

With Killough's method  $S_{ncrt}$  will always lie between  $S_{ncrd}$  and  $S_{ncri}$ . But if the drainage and imbibition curves are made to coincide, the scanning curve will not necessarily follow this combined curve, except at its end points.

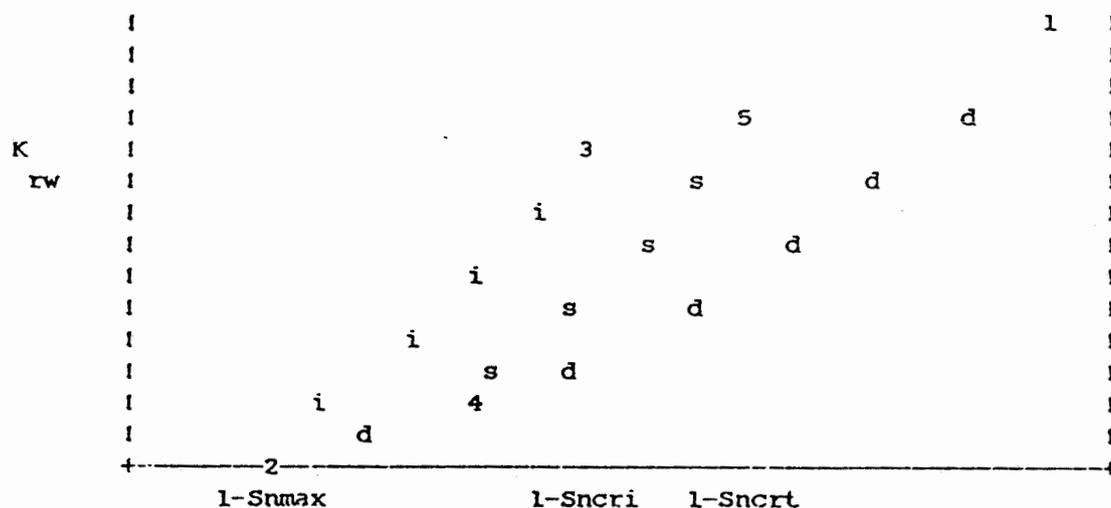
## HYSTERESIS

### Relative Permeability Hysteresis in the Wetting Phase

---

Water is the wetting phase in all runs containing water. Oil is the wetting phase in oil-gas runs. There is an option in two-phase runs to use the Killough model for wetting phase hysteresis. Otherwise the same curve will be used to obtain the wetting phase relative permeability in both drainage and imbibition processes (the user can select either the drainage curve or the imbibition curve). The option is selected in Item 2 of the EHYSTR keyword. It is not available in three-phase runs.

A typical pair of wetting phase relative permeability curves suitable for the Killough model is shown below. The curve marked with d's represents the user-supplied drainage relative permeability table, and the curve marked with i's represents the user-supplied imbibition relative permeability table. The two curves **MUST** meet at the connate saturation ( $S_{wco} = 1 - S_{nmax}$ ). The maximum saturation on the imbibition curve is  $1 - S_{ncri}$ .



Wetting phase saturation----->

An initial drainage process would follow the drainage curve (point 1 to point 2). An imbibition process starting at point 2 ( $S_w = S_{wco} = 1 - S_{nmax}$ ) follows the bounding imbibition curve (point 2 to point 3). Point 3 ( $S_w = 1 - S_{ncri}$ ) is the maximum wetting phase saturation that can be reached starting from  $S_{wco}$ , since the trapped non-wetting phase saturation is  $S_{ncri}$ .

An imbibition process that starts from an intermediate saturation (point 4) will follow a scanning curve (point 4 to point 5). The saturation at point 4 is  $S_w = 1 - S_{hy}$ , where  $S_{hy}$  is the maximum non-wetting phase saturation reached. The maximum saturation that can be reached on the scanning curve

## HYSTERESIS

(point 5) is  $S_w = 1 - S_{ncrt}$ , where  $S_{ncrt}$  is the trapped critical saturation of the non-wetting phase, as defined in the previous section.

If a further drainage process begins from any point on the scanning curve, the same scanning curve is retraced until point 4 is reached, where the drainage curve is rejoined.

Killough's method for calculating the scanning curves uses some of the quantities derived in the previous section for the non-wetting phase. The trapped critical non-wetting phase saturation  $S_{ncrt}$  is determined for the particular value of  $S_{hy}$ . The wetting phase relative permeability at the complementary saturation is calculated, thus fixing the position of point 5,

$$K_{rw}(1-S_{ncrt}) = K_{rd}(1-S_{ncrt}) + (K_{wi}(1-S_{ncri}) - K_{rd}(1-S_{ncri})) [ (S_{ncrt} - S_{ncrd}) / (S_{ncri} - S_{ncrd}) ]^{**A}$$

where the exponent  $A$  is a curvature parameter entered in Item 3 of the keyword EHYSTR.  $K_{rd}$  and  $K_{wi}$  represent the wetting phase relative permeability values on the bounding drainage and imbibition curves respectively.

The relative permeability for a particular saturation  $S_w$  on the scanning curve is

$$K_{rw}(S_w) = K_{rd}(1-S_{hy}) + (K_{rw}(1-S_{ncrt}) - K_{rd}(1-S_{hy})) K_{wi}(1-S_{norm}) / K_{wi}(1-S_{ncri})$$

where  $S_{norm}$  is the function of  $S_n (= 1-S_w)$  defined in the non-wetting phase hysteresis section.

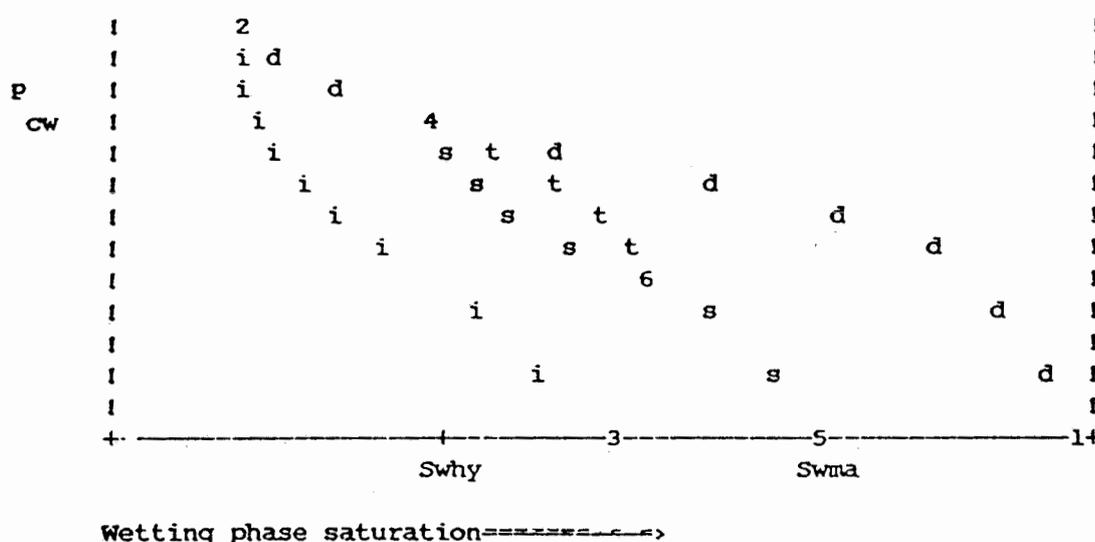
As with Killough's non-wetting phase hysteresis model, if the drainage and imbibition curves are made to coincide the scanning curve will in general only meet this combined curve at its end points (points 4 and 5).

## HYSTERESIS

### Capillary Pressure Hysteresis

---

Given the method described in the previous section for calculating  $S_{ncrt}$ , the trapped non-wetting phase saturation, now consider capillary pressure hysteresis. Again, the data available is the primary drainage and imbibition curves. Consider the water capillary pressure case :



Again, suppose the drainage process starting at 1 is reversed at an intermediate saturation  $S_{hy}$  ( point 4 ). The capillary pressure again traverses a scanning curve from 4 to the trapped critical nonwetting phase saturation at 5. This curve is formed as a weighted average of the drainage and imbibition curves, starting with 100% drainage curve at point 4. The form used is that suggested by Killough :

$$P_c = P_{cd} + F.( P_{ci} - P_{cd} )$$

with

$$F = ( 1/(S_w - S_{why} + E) - 1/E ) / ( 1/(S_{wma} - S_{why} + E) - 1/E )$$

in which :

$E$  is a parameter of the order of 0.1, entered in Item 1 of the EHYSTR keyword.

$S_{why}$  is the water saturation at the hysteresis reversal point 4  
 $S_{wma}$  is the maximum water saturation attainable allowing for the trapped non-wetting phase saturation ( ie at  $S_n = S_{ncrt}$  ).

## HYSTERESIS

Suppose that a second reversal, back to a drainage process, occurs at point 6, rather than continuing to  $S_{wma}$ . Unlike the case of relative permeability hysteresis, this does not re-traverse the scanning curve, but follows a new drainage to imbibition to drainage scanning curve which returns to the point 4, marked by t's in the above diagram. The form of this is :

$$P_c = P_{ci} + G.( P_{cd} - P_{ci} )$$

with

$$G = ( 1/(S_{dep}-S_w+E) - 1/E ) / ( 1/(S_{dep}-S_{why}+E) - 1/E )$$

in which :

$S_{why}$  is the water saturation at the hysteresis reversal point 4  
 $S_{dep}$  is the departure saturation, the point at which  $G = 0$ .

The case in which an imbibition process is reversed on to an imbibition to drainage scanning curve is entirely analogous. The form of the scanning curve is

$$P_c = P_{ci} + F.( P_{cd} - P_{ci} )$$

with

$$F = ( 1/(S_{why}-S_w+E) - 1/E ) / ( 1/(S_{why}-S_{wmax}+E) - 1/E )$$

in which :

$S_{why}$  is the water saturation at the hysteresis reversal point.  
 In this case it is the maximum water saturation in the run.  
 $S_{wmax}$  is the maximum attainable water saturation, which is the same for imbibition and scanning curves

Similar expressions are used for gas capillary pressure hysteresis. In this case the gas saturation is the non-wetting saturation, and drainage to imbibition scanning curves return to the trapped critical gas saturation.

In the three phase and gas water cases both water and gas capillary pressure hysteresis occurs.

## EQUILIBRATION

---

### Equilibration

---

#### 1. Introduction

---

The initial reservoir conditions can be defined in one of three ways:

- (1) They can be read from a restart file generated by an earlier run ( see keyword RESTART ).
- (2) They can be set directly in each grid block, using the keywords PRESSURE, SWAT, SGAS, RS and RV.
- (3) They can be calculated by the equilibration facility.

The equilibration facility is a means of calculating the initial conditions on the basis of hydrostatic equilibrium. If necessary, the reservoir can be divided into separate 'equilibration regions' in which hydrostatic equilibrium exists independently of the other regions. The number of equilibration regions ( NTEQUL ) must be specified in the RUNSPEC section. Within each equilibration region all the grid blocks must use the same pressure table for their PVT properties, but they can use different saturation tables.

#### 2. Data Requirements

---

The data quantities required by the equilibration facility are entered using the keyword EQUIL. For each equilibration region, the engineer should specify the pressure at a given datum depth, the positions of the gas-oil and water-oil contacts, and the value of the capillary pressure at each contact. If there is no gas cap, the gas-oil contact should be placed above the top of the reservoir; and likewise if there is no mobile water, the water-oil contact should be placed far enough below the bottom of the reservoir to exclude the transition zone.

The dissolved gas concentration (  $R_s$  ) can be entered as a function of depth using the keyword RSVD. But this procedure is optional; if a table is not referenced, the dissolved gas concentration in under-saturated oil is everywhere set equal to the saturated  $R_s$  value at the gas-oil contact. If an  $R_s$  v depth table is required for any equilibration region, however, NTEQUL such tables must be entered using the keyword RSVD. The maximum number of depth values in any such table ( NDRXVD ) must be set in the RUNSPEC section. At any position in the reservoir, the  $R_s$  value obtained from an  $R_s$  v depth table is subject to an upper limit equal to the saturated value at the local pressure, since the  $R_s$  value cannot reasonably exceed this.

## EQUILIBRATION

In gas condensate problems, the vapourised oil concentration (  $R_v$  ) in under-saturated gas can be set in a similar manner, using the keyword RVVD. If a table of  $R_v$  v depth is not referenced by a particular equilibration region, the vapourised oil concentration in under-saturated gas is everywhere set equal to the saturated  $R_v$  value at the gas-oil contact. At any position in the reservoir, the  $R_v$  value obtained from a table or the default setting is subject to an upper limit equal to the saturated  $R_v$  value at the local pressure.

The datum depth for the pressure can lie anywhere in the equilibration region, unless the default setting of  $R_s$  or  $R_v$  is required. If in a live oil problem an  $R_s$  v depth table is not referenced for the region, the datum depth must be set at the gas-oil contact. The same restriction is imposed in gas condensate problems whenever an  $R_v$  v depth table is not referenced.

In gas-water problems, instead of a gas-oil contact and a water-oil contact there is a gas-water contact. There is no need to enter any data concerning the non-existent oil phase. But it is also possible to have a gas-water contact in a three-phase problem; namely in a gas condensate problem where all the oil is initially vapourised in the gas phase. In this case the water-oil contact and the gas-oil contact should both be placed at the gas-water contact. In problems containing a gas-water contact, the gas-water capillary pressure should be supplied in the water saturation tables ( keyword SWFN ). For consistency, the capillary pressure data in the gas saturation tables should be set to zero.

### 3. Calculating the Initial Conditions

Within each equilibration region, the calculation is performed in two stages. The first stage sets up an internal table of phase pressures,  $R_s$  and  $R_v$  against depth. The second stage interpolates this table to obtain the fluid conditions in each grid block in the region.

The internal table stores the values of the phase pressures (  $P_o$ ,  $P_w$ ,  $P_g$  ), the dissolved gas concentration (  $R_s$  ), and in gas condensate problems the vapourised oil concentration (  $R_v$  ), at a number of depths within the equilibration region. The depth points are spaced equally throughout the region. The number of depth points in this table ( NDPKVD ) is set in the RUNSPEC section. The default value is 100 , but this can be increased when finer definition is required, for example in problems where the oil zone occupies only a small fraction of the overall reservoir thickness.

Between each pair of depth points in the table, the pressure gradient of each phase is calculated iteratively using a density consistent with the average pressure within the depth step. If the datum depth lies in the oil zone, the oil pressure values are calculated first, stepping from the datum depth up to the top and down to the bottom of the equilibration region. The water pressure on the oil-water contact can now be obtained, and the water

## EQUILIBRATION

pressure values are calculated. Similarly the gas pressure values are calculated starting from the gas-oil contact. The  $R_s$  values are calculated, either from an  $R_s$  v depth table supplied in the input data or by defaulting to the saturated  $R_s$  value on the gas-oil contact. The values are subject to an upper limit equal to the saturated  $R_s$  value at the local pressure. The  $R_v$  values are obtained in a similar manner in gas condensate problems.

In the second stage of the equilibration calculation, the local fluid conditions are determined in each grid block in the equilibration region. The internal table is interpolated to obtain the values of  $P_o$ ,  $P_w$ ,  $P_g$ ,  $R_s$  and  $R_v$  at the grid block centre depth. The water saturation is determined by inverse look-up of the water capillary pressure table ( entered with keyword SWFN ) for the grid block, such that

$$P_{cow}( S_w ) = P_o - P_w$$

If  $P_o - P_w$  exceeds the highest capillary pressure value in the SWFN table ( corresponding to the lowest saturation value  $S_{wmin}$  ), the water saturation is set equal to  $S_{wmin}$ . If  $P_o - P_w$  is less than the lowest capillary pressure in the SWFN table ( corresponding to the highest saturation value  $S_{wmax}$  ), the water saturation is set equal to  $S_{wmax}$  and the oil pressure is adjusted to follow the water pressure gradient.

The gas saturation is similarly determined by inverse look-up of the gas capillary pressure table ( keyword SGFN ) for the grid block, such that

$$P_{cog}( S_g ) = P_g - P_o$$

If  $P_g - P_o$  is less than the lowest capillary pressure value in the SGFN table ( corresponding to the lowest saturation value  $S_{gmin}$  ), the gas saturation is set equal to  $S_{gmin}$ . If  $P_g - P_o$  exceeds the highest capillary pressure in the SGFN table ( corresponding to the highest saturation value  $S_{gmax}$  ), the gas saturation is set equal to  $S_{gmax}$  and the oil pressure is adjusted to follow the gas pressure gradient.

In live oil problems, the  $R_s$  value is simply interpolated from the internal table at the grid block depth. However, if there is a non-zero gas saturation at that depth the  $R_s$  value is re-set to the saturated  $R_s$  value at the local pressure. The  $R_v$  value is obtained in a similar manner in gas condensate problems.

Since the water and gas saturations outside the transition zones are set to the respective end-point values in the SWFN and SGFN tables, the end-points of the two tables must be consistent. Normally the lowest gas saturation in the SGFN table should be zero ( so that  $S_g = 0$  below the gas transition zone ), and the highest gas saturation in this table should be  $1 - S_{wmin}$  ( so that  $S_o = 0$  above the gas transition zone ). The lowest water saturation value in the SWFN table should be the connate value ( so that  $S_w = S_{wco}$  above the water

## EQUILIBRATION

transition zone ), and the highest water saturation in this table should be 1.0 ( so that  $S_w = 1.0$  below the water transition zone ). The consistency requirements between saturation tables are detailed in the Technical Appendix entitled 'Saturation Functions'.

In cases where the water-oil and gas-oil transition zones closely overlap, the treatment outlined above may produce negative oil saturations. For example, where the water-oil transition zone extends above the gas-oil transition zone, the water saturation (  $S_w > S_{wmin}$  ) and the gas saturation (  $S_g = S_{gmax}$  ) will add to more than 1.0 . If this occurs, the water and gas saturations will be recalculated from the gas-water capillary pressure, which is taken as the sum of the water-oil and gas-oil capillary pressures

$$P_{cow}( S_w ) + P_{cog}( S_g = 1 - S_w ) = P_g - P_w$$

### 4. Accurate Fluids in Place Calculation

The accuracy of the initial fluids in place calculation can be controlled by the engineer. There are three options, which are selected according to the value of an integer (  $N$  ) specified in the EQUIL keyword.

(1) Centre-point equilibration (  $N = 0$  ).

The simulator sets the fluid saturations in each grid block according to the conditions at the centre of the block. This option is the fastest of the three, but it is the least accurate, particularly in cases where a fluid contact passes through large grid blocks.

(2) Horizontal block averaging (  $N < 0$  ).

The top and bottom halves of each grid block are divided into a specified number of equally spaced layers, and the saturations are determined locally in each layer. The phase saturations for the block are set equal to the average of the saturations in each layer. This option provides a more accurate calculation of the fluids in place.

(3) Tilted block averaging (  $N > 0$  ).

This option is similar to option (2), but it takes into account the slope of each grid block. The top and bottom faces of the blocks are treated as planes that are tilted about their central points. If the NEWTRAN keyword has been entered, the tilt is determined from the block corner point depths. Otherwise the faces are tilted towards the centres of the corresponding faces of the neighbouring blocks within the equilibration region. The phase saturations in each block are calculated as a weighted average of the saturations in each layer, weighted according to the area of each layer that is enclosed within the block. This option provides the most accurate

## EQUILIBRATION

calculation of fluids in place.

Option (1) produces 'quiescent' initial conditions, making a true steady-state solution. The more accurate options (2) and (3), however, do not produce absolutely quiescent initial conditions. There will be a redistribution of fluids between grid blocks near the contacts when the simulation begins, which occurs independently of any external driving force ( wells etc. ). The reason is that a steady state solution on the fine equilibration grid ( in which each block is subdivided into several layers ) is not necessarily a steady state solution on the coarser simulation grid. If the redistribution of fluids produces a significant transient when the simulation is started, the engineer can overcome this by setting the 'quiescence switch' ( QUIESC in the RUNSPEC section ). If this switch is set to T, modifications will be applied to the phase pressures to make the initial solution a true steady state. These pressure modifications will be applied for the duration of the run.

If the Vertical Equilibrium option is used ( QVEOPT = T in the RUNSPEC section ), an exact VE equilibration method can be chosen by selecting option (1). Instead of the centre-point calculation, the option will calculate the fluid saturations in each grid block directly from the contact depths. It employs the same relationship between saturations and contact depths that is subsequently used for the simulation. Option (1) is the recommended option in VE runs.

### 5. The Threshold Pressure Facility

If the reservoir contains two or more communicating equilibration regions having different pressures or contact depths, fluid will flow between these regions immediately the simulation begins. The 'quiescence switch' will only produce local steady state conditions within each separate equilibration region. Global quiescence, however, can be achieved by using the Threshold Pressure facility. The facility prevents flow between adjacent equilibration regions from occurring until the potential difference exceeds a specified threshold value. Thereafter the potential difference for flow across equilibration region boundaries is reduced by the appropriate threshold value.

The Threshold Pressure facility is enabled by setting the switch QTHPRS to T in the RUNSPEC section. Each interface between adjacent equilibration regions will then have its own threshold pressure. If QREVTH is also set to F in the RUNSPEC section, each interface will have two threshold pressures, one for flow in each direction. The threshold pressure values are set using the keyword THPRES. Values can be defaulted to the maximum potential difference between communicating cells across the interface ( i.e. just sufficient to prevent flow occurring across the interface before the reservoir is perturbed ).

## 7.0 CONSTITUENT TRACKING

The ECLIPSE model includes a constituent tracking by convection (and numerical diffusion) only. This is an approximate method for determining constituent transport.

Also, the model oil properties to vary regionally. These two options are described in this section.

## TRACER TRACKING

---

### Tracer Tracking

---

The ECLIPSE tracer tracking option is a general facility to follow the movement of 'marked' fluid elements during a simulation run.

The tracer tracking option has a wide variety of reservoir modelling applications. In the case of tracers defined to exist in the water phase, it may be used, for example, to determine the movement within the reservoir of water injected into any number of injection wells or to predict the variations in salinity or concentration of other chemical species in the water produced from the reservoir.

Tracers may also be defined to exist in a hydrocarbon phase, and the ECLIPSE tracer option will model the exchange of the tracer fluid elements between the free and solution states whenever mass transfer takes place in the associated hydrocarbon phase. The tracer option is therefore useful in predicting the movement of the initial free gas or solution gas during the evolution of reservoir.

The tracer concentrations can be initialised on a region basis using a depth table method of data input. This method of data input is especially suited to tracking fluid which initially exists within a particular region of the reservoir. In the case of gas condensate reservoirs, it is thus possible to track the oil which initially exists in the vapour phase in separate regions of the reservoir and to determine its subsequent movement through the reservoir grid. The tracer option could also be used to predict the concentration of chemical impurity species ( such as the sulphur content of the oil ) in the hydrocarbon production streams.

The ECLIPSE tracer tracking option allows up to 50 tracers to be defined in a single run. Each tracer exists in a particular stock tank phase although mass exchange of the tracer fluid may occur between the free and solution states of each phase. This ensures that if, for example, a tracer is defined to be the dry gas injected into an undersaturated oil reservoir the solution of the injected tracer gas into the oil is modelled accurately. Re-injection of tracer gas cannot, however, be modelled in the current tracer option.

The current ECLIPSE tracer tracking option assumes that the presence of tracers does not affect the PVT properties of the phases in which they are embedded. The tracers are thus to be regarded as passive. The tracer equations are a set of conservation equations for each tracer species with an exchange term for hydrocarbon phase tracers to model mass transfer effects. The tracer concentrations are updated fully-implicitly at the end of each time-step after the oil, water and gas flows have been computed.

## TRACER TRACKING

### Control Of Numerical Diffusion

---

The differential equations governing the evolution of the concentration of tracer species within each grid cell are a series of linear convection equations. ECLIPSE solves the tracer transport equations at each time-step using a fully implicit time discretisation. Such a system of difference equations is particularly prone to exhibiting the effects of numerical diffusion errors, particularly in regions of the grid where tracer concentrations vary most rapidly. The smearing of tracer interfaces tends to be most apparent in single phase regions ( e.g. gas injection into a gas cap, water injection into aquifers ).

An option has been included in the ECLIPSE tracer model to reduce the effects of numerical diffusion on tracer interfaces. At present the option applies to tracers embedded in phases where no mass transfer effects can take place. The option is activated in tracer runs by means of the QDISP parameter in the RUNSPEC section of the input data file.

### Using the ECLIPSE Tracer Tracking Facility

---

The tracer tracking facility in ECLIPSE is activated by means of three integers NOTRAC, NWTRAC, NGTRAC set in the RUNSPEC section of the input data. These three integers represent the maximum number of tracers to be defined in each of the three stock tank phases - water, oil and dry gas. It is recommended that the precise number of tracers to be tracked should be specified in the RUNSPEC section in order to minimise the memory requirements of the model.

The names of each tracer and their associated stock tank phases are defined using the TRACER keyword in the PROPS section of the input data. The maximum number of rows of data is (NOTRAC+NWTRAC+NGTRAC). Each tracer name is, at present, limited to be a name of up to 3 characters in length. The output of TRACER data is controlled by the 10th. argument in the RPTPROPS keyword.

In the case of tracers defined to exist in the gas phase when the QDIGAS parameter (see RUNSPEC section) is set to T, or tracers defined in the oil phase when the QGCOND parameter (see RUNSPEC section) is set to T, then the full tracer solution consists of a concentration in the free and solution state of its associated phase. ECLIPSE automatically allocates space for tracers which may exist in the free or solution state, and issues error messages whenever insufficient data is provided to calculate the initial tracer conditions.

The initial tracer concentrations may be supplied in one of two ways. The most direct method is enumeration using the keyword TBLK in the SOLUTION section of the input data. An alternative method is to define a series of

## TRACER TRACKING

tracer concentration versus depth tables in the SOLUTION data, together with the associated region numbers in the REGIONS section of the input data. The data is entered by means of the keywords TVDP and TNUM respectively, and control of the depth table and region data output is achieved using the RPTSOL and RPTREGS keywords. The keywords TBIK, TNUM and TVDP must be used in conjunction with the character F or S ( free or solution ), together with the tracer name, to make a composite keyword of up to 8 characters in length. For example, to initialise a water phase tracer whose name is 'WTR', the keywords required for the region/depth table method of input would be TVDPFWTR and TNUMFWTR.

The user may require to specify the initial concentration of a water phase tracer in each of the analytic aquifers in a simulation run. This can be achieved using the AQANTRC keyword in the SOLUTION section of the input data. Output of the status of each water tracer for each analytic aquifer is obtained at each report time using item 13 of the RPTSCHED keyword.

Tracer injection control is achieved by means of the WTRACER keyword in the SCHEDULE section. This keyword allows the specification of the concentration of a particular tracer in the injection stream for each well. A default injection concentration value of zero is assumed for tracers not defined under the WTRACER keyword.

Reporting of the tracer concentration in each grid cell is activated using the 32nd argument of the RPTSCHED keyword. Output of tracer volumes within each reporting region and the inter-region tracer flows are available using the 33rd keyword argument of the RPTSCHED keyword. Grid block arrays of tracer concentration are produced under a mnemonic consisting of the tracer name together with the character F and/or S. For example, if IGS is a gas phase tracer in a run with dissolved gas ( QDIGAS = T in the RUNSPEC section ), then two grid block concentration arrays are produced under the mnemonics IGSF and IGSS which represent the local concentration of IGS in the free and dissolved gas respectively.

Completion, well, group and field reports for production/injection rates and cumulative production/injection of each tracer are produced using levels 1-5 of the 9th. argument of the RPTSCHED keyword.

## API TRACKING

---

### API Tracking

---

The API Tracking facility enables ECLIPSE to model the mixing of different types of oil, having different surface densities and PVT properties.

Without the API Tracking facility, the presence of different types of oil in the reservoir could be handled with the aid of PVT region numbers. Oil in PVT region 1 would have its properties determined from PVT table number 1, and so on. However, this method cannot model the mixing of oil types. Oil flowing from region 1 into region 2 would appear to take on the properties associated with region 2.

The API Tracking facility essentially replaces the concept of PVT regions for oil. The PVT tables used for determining the oil properties are selected at each time step according to the average API of the oil in each grid block (or to be more precise, its average surface density). A mass conservation equation is solved at the end of each time step to update the oil surface density in each grid block, to model the mixing of the different oil types.

The API Tracking facility is turned on with the switch QAPITR in Record 6 of the RUNSPEC section.

### The PVT Properties

Two or more sets of PVT tables should be supplied, each set being associated with a particular value of the API. The oil property tables are entered in the usual way (with keyword PVTO for live oil problems, or keyword PVDO for dead oil problems), but with the requirement that all the PVTO tables must have the same set of  $R_s$  nodal values. This restriction allows ECLIPSE to use an efficient method of interpolating between tables, for intermediate values of the API. The bubble point pressures, formation volume factors and viscosities may of course vary from table to table, to reflect the different PVT properties of the different types of oil.

The API values associated with each PVT table are set using the keyword GRAVITY, which inputs the oil, water and gas gravities for each PVT table number. The API values are converted internally to oil surface densities, using the formula given in the Appendix on Unit Conventions. The oil API values must DECREASE monotonically with table number. Alternatively the surface densities associated with each table number can be input directly using the keyword DENSITY. In this case the oil surface densities must INCREASE monotonically with table number.

The PVT properties of water and gas are not affected by the API tracking

UNIT CONVENTIONS

-----  
 Unit Conventions  
 -----

There are three unit conventions: METRIC units  
 FIELD units  
 LAB units

The LAB convention is intended for use when simulating laboratory-scale experiments. It is based on the original Darcy units, except that the unit of time is the hour rather than the second.

The units for each data quantity are given in the table below.

<u>Quantity</u>	<u>METRIC</u>	<u>FIELD</u>	<u>LAB</u>
Length, Depth, Radius	METRES	FEET	CM
Time	DAYS	DAYS	HOURS
Density	KG/M3	LB/FT3	GM/CC
Pressure ( absolute )	BARS	PSIA	ATMOSA
Pressure difference	BARS	PSI	ATMOS
Compressibility	1/BARS	1/PSI	1/ATMOS
Viscosity	CPOISE	CPOISE	CPOISE
Permeability	MDARCY	MDARCY	MDARCY
Liquid volume ( surface conditions )	SM3	STB	SCC
Gas volume ( surface conditions )	SM3	MSCF	SCC
Fluid volume ( reservoir conditions )	RM3	RB	RC
Liquid rate ( surface conditions )	SM3/DAY	STB/D	SCC/D
Gas rate ( surface conditions )	SM3/DAY	MSCF/D	SCC/D
Fluid rate ( reservoir conditions )	RM3/DAY	RB/D	RC/D
Formation volume factor ( liquid )	RM3	RB	RC
Formation volume factor ( gas )	RM3	RB	RC
Gas-oil ratio			
Oil-gas ratio	SM3/SI		
Transmissibility	CP-M3,		
Productivity index ( liquid )	SM3/D,		
Productivity index ( gas )	SM3/D,		
D-factor	DAY/S		
Temperature	DEG C		
Angle	DEGRE		

## API TRACKING

option. The same number of tables will have to be entered in all the PVT data keywords and the ROCK keyword, but normally only the first table will actually be used in all except the oil PVT data. The unused PVT and rock compressibility tables can be defaulted by typing a single slash ( / ) on a new line for each unused table in the keyword.

The API tracking facility over-rides the concept of PVT table regions for oil. The tables used to determine the oil PVT properties are selected according to the surface density of the oil in the grid block, instead of the block's PVT region number. Thus the PVTNUM keyword in the REGIONS section should not normally be used with the API tracking facility. If necessary, however, PVT region numbers can still be used to model different types of gas or water, although the mixing of these different fluids will not be handled properly.

### Setting the Initial Conditions

The initial API distribution throughout the reservoir is specified in the SOLUTION section. If the initial conditions are determined by equilibration, the API can vary with depth independently in each equilibration region. Keyword APIVD supplies a table of API values vs. depth for each equilibration region.

When setting the initial conditions by enumeration (keywords PRESSURE, SWAT etc.), the keyword OILAPI is used to supply the initial API values for each grid block.

### The API Tracking Calculation

The initial API values in the grid blocks are immediately converted into oil surface density. The hydrostatic pressure gradient calculation takes account of the varying surface density of the oil. The variation of other PVT properties with surface density (bubble point pressure, formation volume factor and viscosity) is handled by interpolating between PVT tables. The two adjacent PVT tables whose oil surface densities straddle the oil surface density value in the grid block are located, and their properties (  $1/B_o$  ,  $1/(B_o v_o)$  ) are interpolated linearly in oil surface density.

The oil surface density in each grid block and well is held constant over the time step. When a converged solution for the time step has been found, and the inter-block flows determined, a mass conservation equation is solved to update the oil surface densities. The new densities will be used in the next time step.

The oil surface densities are converted back into API values for the output reports. The 31'st switch in the RPTSCHED keyword provides an output of the grid block API values, and the well reports include the API values in the wells.

## 8.0 UNIT CONVENTIONS

The user options for units are listed in this section.

UNIT CONVENTIONS

-----  
 Unit Conventions  
 -----

There are three unit conventions: METRIC units  
 FIELD units  
 LAB units

The LAB convention is intended for use when simulating laboratory-scale experiments. It is based on the original Darcy units, except that the unit of time is the hour rather than the second.

The units for each data quantity are given in the table below.

<u>Quantity</u>	<u>METRIC</u>	<u>FIELD</u>	<u>LAB</u>
Length, Depth, Radius	METRES	FEET	CM
Time	DAYS	DAYS	HOURS
Density	KG/M3	LB/FT3	GM/CC
Pressure ( absolute )	BARSA	PSIA	ATMOSA
Pressure difference	BARS	PSI	ATMOS
Compressibility	1/BARS	1/PSI	1/ATMOS
Viscosity	CPOISE	CPOISE	CPOISE
Permeability	MDARCY	MDARCY	MDARCY
Liquid volume ( surface conditions )	SM3	STB	SCC
Gas volume ( surface conditions )	SM3	MSCF	SCC
Fluid volume ( reservoir conditions )	RM3	RB	RC
Liquid rate ( surface conditions )	SM3/DAY	STB/D	SCC/D
Gas rate ( surface conditions )	SM3/DAY	MSCF/D	SCC/D
Fluid rate ( reservoir conditions )	RM3/DAY	RB/D	RC/D
Formation volume factor ( liquid )	RM3	RB	RC
Formation volume factor ( gas )	RM3	RB	RC
Gas-oil ratio			
Oil-gas ratio	SM3/SI		
Transmissibility	CP-M3,		
Productivity index ( liquid )	SM3/D,		
Productivity index ( gas )	SM3/D,		
D-factor	DAY/S		
Temperature	DEG C		
Angle	DEGRE		