

Special Purpose Reservoir Simulators-Overview

Introduction

Black oil models encompass as much as 80% of all reservoir simulation applications.

But there may be times when we find ourselves dealing with totally different physical approach.

Water Coning Simulators

Water coning, a premature invasion of the wellbore by water, usually occurs in bottom-drive reservoirs.

The oil-water contact, which is normally horizontal, becomes distorted near the wellbore, assuming an upward concave posture.

Coning usually occurs if the well is completed close to the oil-water contact and produced at a high rate, creating a sharp pressure gradient near the wellbore and resulting in excessive water production.

This water invades the pore spaces around the perforations inhibits the flow of oil towards the wellbore.

A similar phenomenon often takes place near the gas-oil contact where the gas cap becomes distorted as the gas cusps downward and causes excessive gas production



When using a reservoir simulator to study water coning or gas cusping, we must take extra care to capture the rapid changes that take place within the immediate vicinity of the wellbore.

To enhance the description of these near-wellbore changes, we resort to single-well modeling most of the time.

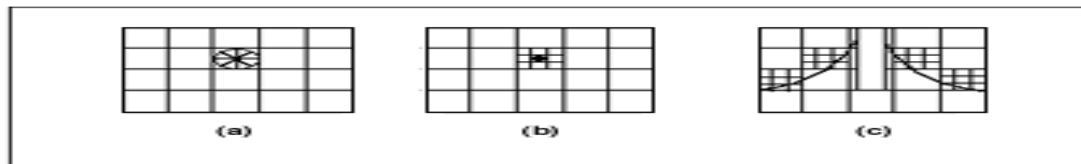
A radial-cylindrical grid with smaller grid spacing along the radial-direction and dense spacing along the vertical direction, captures the movement of the saturation front.



- ▶ While the data requirement for a coning study is the same as for a full field model, the grid system is significantly different.

Sometimes, we superimpose rectangular geometry with a hybrid grid system over the computational domain (part a)

- ▶ In some cases, if simulators can handle locally refined grids, then we can use local refinement techniques effectively. We can place these locally refined sections around the wellbore (part b), or they can be dynamic such that they move together with the oil-water contact (part c).



Dual Porosity/Permeability Simulators

- ▶ With more and more reservoirs being identified as naturally fractured, there is a growing interest in simulating such systems.
- ▶ We generally characterize naturally fractured reservoirs as *dual porosity* systems because of the presence of two continua: the rock matrix and the fractures.
- ▶ We usually represent these continua using two different sets of porosity and permeability functions, referring to the matrix porosity as *primary* and the fracture porosity as *secondary*.

Fluids are stored mainly in the rock matrix, and transmitted through the fracture network; the permeability of the fracture network is much higher than that of the matrix.

Porosity and permeability are not the only discontinuous functions in a dual-porosity system.

Other properties, such as the capillary pressure and relative permeability functions, also differ as we move from one subdomain of flow to the other.

There are two general approaches to simulating dual-porosity systems:
dual-porosity/single-permeability and dual-porosity/dual-permeability.

In the *dual-porosity/single-permeability approach*, we write flow equations for only one of the two continua i.e the fracture network.

We account for matrix flow by imposing a source term on the flow equations for the fracture continuum.

We calculate this mass transfer term between the matrix blocks and the fractures by using a transfer function, which is based on either unsteady-state or pseudo-steady-state computations.

In the *dual-porosity/dual-permeability approach*, we write flow equations for both continua.

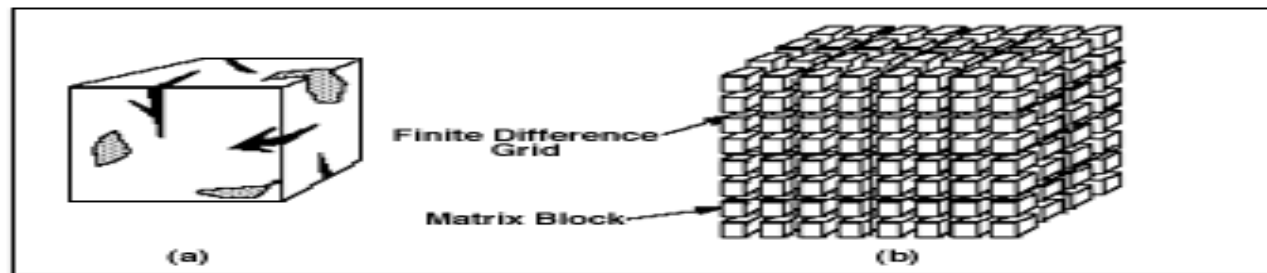
We subdivide matrix blocks into grid blocks, and solve flow equations within each of these subdomains (matrix blocks).

Again, we calculate the transfer of fluid between the matrix and the fracture using the existing pressure gradients at the interface.

Natural fractures occur randomly in the porous medium (part a). However, in simulating the fracture system, we idealize the system as shown in part b.

A computational block consists of several matrix blocks.

In part b, each computational block is made up of eight matrix blocks.



Although the matrix blocks in the idealized dual-porosity system (part b) are shown as cubes, we could have shown them as spheres, cylinders, or slab elements.

Studies have shown that there is no significant disparity between results obtained by using any of these geometrical configurations.

The main difference usually results from the handling of the matrix/fracture fluid exchange and the types of the capillary pressure and relative permeability functions used within the fracture network.

Thermal Recovery Simulators

Thermal recovery processes are designed to raise the temperature of reservoir oil, thereby decreasing its viscosity and enhancing its flow characteristics.

The primary differences among various thermal recovery methods are in the *heat sources used to raise oil temperature.*

The two most popular methods are steam flooding and in-situ combustion.

Of all enhanced oil recovery (EOR) processes, steam flooding is among the most successful.

The major difference between thermal recovery simulators and other types of models is the need for the energy balance equation.

Temperature distribution is the main driving factor in thermal recovery, and so must be adequately predicted--especially since viscosity is a strong function of temperature.

The energy equations are usually highly nonlinear and strongly related to the mass balance equations.

In addition, two types of heat transfer mechanisms are accounted for:
conduction and convection.

The behavior of the energy equation depends on which of the two mechanisms dominates.

If conduction is dominant, the equation exhibits parabolic behavior (diffusivity equation) but if convection dominates, it exhibits hyperbolic behavior (shock wave equation).

This translates to sharp thermal fronts, which leads to a numerical smearing of the front. In this case, we must pay special attention to the numerical scheme in order to avoid numerical instability.

One other feature of thermal simulators is the need to calculate the heat loss to the surrounding formations.

Whereas in formulating fluid flow equations, we consider the reservoir an isolated system with no fluid exchange with the surrounding, thermal equations allow heat exchange with the surroundings, at least by conduction.

This is more critical for the overburden and under-burden because of the large contact area relative to the adjacent formations in the lateral directions.

Steam flooding enhances oil recovery by transferring heat from the steam to increase the temperature of the reservoir section adjacent to the wellbore.

This temperature increase reduces the oil viscosity and hence the flow resistance in the wellbore vicinity.

There are basically two types of steam injection schemes: *cyclic*, where injection and production are done sequentially using the same well, and *continuous*, which involves separate injectors and producers.

The primary effects being simulated in steam injection are the temperature increase and the resultant decrease in the oil viscosity.

There are two basic types of steam flooding simulators: compositional and non-compositional.

Non-compositional simulators are simpler and usually adequate for most cases.

But in cases where we suspect that the distillation of light components significantly affects the stimulation process, we should use a compositional steam flood simulator.

Such a simulator requires a phase behavior description for the oil/steam system.

An additional complication arises when three-phase relative permeability data are considered temperature-dependent.

The main computational challenges arise from the strong inherent nonlinearities in the energy equations, the discontinuity resulting from phase changes (condensation, vaporization), and a strong coupling between the fluid movement and the energy transfer.

These peculiar characteristics manifest themselves in numerical problems such as instability and grid orientation effects.

In-situ combustion uses basically the same principle as steam injection, i.e., using heat to reduce viscosity of oil.

In this case, however, the heat comes from injecting air into the reservoir and igniting part of the oil to start a combustion front.

The primary consideration in simulating in-situ combustion, apart from all the other factors mentioned in the discussion of steam flooding, is the combustion reaction kinetics.

The temperature dependence for the kinetics equations is usually described by Arrhenius type rate expressions.

These strongly temperature-dependent functions introduce a new level of nonlinearity to the energy equations.

We must be aware of possible severe stability problems in in-situ combustion simulators

Compositional Simulators

Compositional reservoir simulators account for multiphase flow and interfacial mass transfer of each component in a hydrocarbon system.

This implies that at any given time, the simulator tracks fluid movement and establishes the state of equilibrium of the reservoir fluids at the discrete points.

At each node, phase pressure, phase saturation and overall composition are computed as a function of time.

Compositional simulators are particularly useful in describing gas condensate reservoirs, volatile oil reservoirs, gas cycling processes and in some thermal recovery processes, in which compositional changes are important

The distinguishing feature of a compositional simulator is its full coupling of the fluid phase behavior model with the flow equations and perhaps the energy equation.

In this case, rigorous flash calculations are performed using either tabular equilibrium ratios or analytical equations of state.

All the volumetric and thermal properties are computed as being composition-dependent.

There are certain limitations inherent in using compositional reservoir simulators.

The two principal ones are the excessive CPU time requirements and the problem of adequately describing the fluid phase behavior.

In compositional simulation, we generally treat petroleum as a mixture of limited discrete components.

But in fact, it is a *continuous* mixture. The standard method of handling this problem is the use of C7+.

However, difficulties often arise from assigning the needed parameters to this pseudo-component.

Miscible Displacement Simulators

A miscible displacement process involves two or more fluids that are mutually miscible when they come into contact in all proportions.

When complete miscibility takes place, there is no interface formed between components.

The miscibility between two components can take place in two different ways: *first-contact miscibility and multiple-contact miscibility.*

In the first case, the displacing fluid is immediately miscible with the displaced fluid, while in the second, miscibility occurs after a series of equilibrium contact stages.

Examples of miscible displacement processes include chemical flooding (e.g., miscible carbon dioxide injection, polymer flooding, micellar flooding) and displacement of oil by solvents.

Miscible displacement simulators can be *multi-component and multi-mechanistic models*.

Multi-mechanism indicates that flow is taking place due to convection and dispersion.

We can represent *multi-mechanistic flow* by describing the velocity of a component as the sum of the velocities due to different mechanisms.

In a miscible displacement simulator, we generally assume single-phase flow.

This assumption implies the presence of full miscibility and lets us avoid difficult vapor-liquid equilibrium computations of multiple-contact miscibility.

Furthermore, we usually consider two components (possibly oil and the solvent), and emphasize flow by dispersion in the construction of these simulators.

We calculate phase properties such as viscosity and density using mixing rules. This type of formulation assumes no volume change upon mixing.

Chemical and Polymer Flooding Simulators

Chemical flooding simulators are much more demanding to develop than other special purpose simulators.

This is simply because the physics involved in a typical chemical flood are much more complex, and require consideration of the extensive microscopic phenomena that are taking place at the fluid-fluid and fluid-rock interfaces.

Chemical floods employ several different fluids, and therefore form several fluid banks.

Interfacial phenomena, phase behavior of the complex systems, adsorption and desorption of certain chemical agents to and from the rock grains make the problem even more complicated.

Most chemical flood simulators are developed to study certain phenomena in the laboratory, where it is much easier to control process variables.

***Polymer injection* is a complex process.**

It involves simultaneous multiphase fluid flow with interphase mass transfer of water between the polymer and water phases when there is adsorption of polymer on the rock of the porous medium.

In a polymer injection simulator, the polymer and water phases form the aqueous phase.

Usually water is allowed to transfer from the polymer phase to the water phase as a result of polymer slug deterioration caused by polymer adsorption on rock.

- ▶ **However, most of the time, water is not allowed to transfer from the water phase to the polymer phase (i.e., the polymer slug cannot be diluted by water).**
-
- ▶ **It is necessary to consider the polymer adsorption on the rock and its effects on the permeability of the rock to the existing phases.**
 - ▶ **We also need to account for transverse dispersion of the polymer component within the aqueous phase.**
 - ▶ **Most polymer injection simulators treat the adsorption of polymer on the rock as permanent (i.e., there is no desorption).**

We can use chemical and polymer flood simulators, like other simulators, as screening tools to select optimal patterns.

We can also use them to determine optimal slug sizes, and to analyze the increased production and profitability of a full scale chemical flood under several operating strategies.

Finally, we can use them to predict the effect of fluid and rock properties on the oil recovery and flood performance.

Coalbed Reservoir Simulators

- ▶ **The petroleum industry classifies methane from coalbed reservoirs as unconventional natural gas.**
- ▶ **Unconventional resources offer significant potential, both now and in the future, in terms of large volumes of reserves, low production costs, and relatively simple development techniques.**

One of the more important characteristics of coal seams is their dual-porosity nature, characterized by *well-defined macro-pore and micro-pore structures*.

The natural fracture networks of coal seams are uniformly distributed, and composed of two fracture systems which are almost orthogonal to each other (face and butt cleats).

While the face cleat is continuous throughout the reservoir and capable of draining large areas, the butt cleat is discontinuous and ends at the face cleat.

Thus, the anisotropic nature of a coal seam originates from this cleat system in which the permeability in the direction of a face cleat is considerably larger than that in the direction of a butt cleat.

The micro-pore system, as a primary porosity matrix, has a size in the order of molecular dimensions to a few Angstroms.

In general, we assume that the openings are not accessible to water, and that they contribute the major portion of gas storage in areas in which gas is stored in adsorbed and free states.

Quantitatively, as much as 2000 SCF of methane can be stored in a ton of coal by adsorption.

When the coal seam is in virgin conditions, the volume of the free gas in the micro-pores is almost negligible compared to the volume of the gas in the adsorbed state.

Again, for coal seams in virgin conditions, we assume that the cleat system is fully saturated with water.

As water is removed from the macro-pores, gas is desorbed from the micro-pore surfaces toward the macro-pore structure.

This process represents a distributed source mechanism over the macro-pore structure.

As the desorption process continues, the free gas saturation within the fracture network increases.

Most coalbed simulators use Langmuir sorption isotherms to describe the release of methane from the adsorbed state. They achieve this by solving a first order kinetic sorption model.

Along these lines, *two approaches have been developed*: equilibrium sorption isotherms (pressure dependent) and non-equilibrium sorption isotherms (pressure and time dependent).

In the ***equilibrium sorption*** isotherm approach, we assume that the gas adsorbed onto the micro-pore walls is in a constant state of equilibrium with the free gas phase in the macro-pore system.

Models based on this approach are essentially single-porosity models altered for coal seams by either the inclusion of a pressure dependent source term or by the modification of the storage term.

These models generally predict optimistic results, since the adsorbed gas is assumed to instantaneously enter the macro-pore system.

The overall structure of coalbed reservoir simulators is similar to the conventional dual-porosity/single-permeability models.

With the aid of these models, production performances of coalbed reservoirs have been thoroughly examined.

If the simulator has the options to accommodate ongoing mining activities, it can be used to predict the methane emission rates into the active mine working area.

The first-generation coalbed reservoir models consider the only existence of a single component gas (methane) and water.

In some coalbed reservoirs, components other than methane, (e.g., such as carbon dioxide), may play an important role in the coal's sorption characteristics.

Similarly, enhanced recovery of coalbed methane through nitrogen or carbon dioxide injection is under consideration as a viable process to increase the rate of methane recovery.

In studying these types of applications, it will be necessary to use *second-generation coalbed reservoir models* which use a compositional approach in modeling the selective adsorption/desorption of different gas components.