Simulation

To simulate fluid flow in a reservoir, we first need a 3D or 4D geological model of the reservoir. This model will have to be described in such a way that the data can be used for numerical computations. This is done by "gridding" the reservoir, which divides the reservoir into a finite set of homogeneous grid cells. Each of these cells contains data for each of the geological parameters in that cell.

Next we need a description of the fluids in the reservoir. This is done by describing the so-called relative permeability functions and the PVT data. Relative permeability functions are a description of how the fluids flow relative to each other. The PVT data represent how pressure and volume of the fluids relate. We also have to describe where the fluid phases are present in the reservoir – this is done by describing where their contacts are sitting.

We need to apply a force to the fluids and this is done by setting locations of wells and their production/injection rates. Lastly, we must specify the time period for the simulation and how many time steps we would like to use.

The flow of fluids is governed by three basic physical principles:

- Mass is neither created nor destroyed, that is Conservation of Mass.
- The rate of change of momentum of a portion of the fluid equals the force applied to it, or Conservation of Momentum.
- Energy is neither created nor destroyed, that is Conservation of Energy.

Conservation of Mass

The first principle is modeled by a so called partial differential equation that says that if there is a difference between flow into and out of a tiny volume of space, this will either cause a build-up or a drawdown of mass in this tiny volume.

Conservation of Momentum

The second principle is approximated by an experimental law, called Darcy's law, that relate the pressure difference (force) across a porous rock containing a fluid and the resulting velocity (momentum) of this fluid. This experimental law is also represented by a partial differential equation that is combined with the equation based on the first principle to form a set of partial differential equations that is the mathematical description of the flow of fluids through a porous media.

Conservation of Energy

The third principle is approximated by a relationship between Pressure, Volume and Temperature (PVT) for the rock and the fluids. The PVT data is represented by a set of tables for each reservoir.

General Information on Streamline Simulation

FrontSim is a streamline simulator. This numerical method first solves the pressure, then computes the streamlines and thereafter computes the changes in saturation along the streamlines. The equations for saturations are solved as several one dimensional problems along streamlines, as opposed to solving one large three dimensional problem. Consequently, FrontSim can be very fast and hence capable of handling grids with millions of grid cells.

For reservoirs where the movement of the fluids is mostly driven by the potential field induced by producing wells, FrontSim is particularly efficient. Examples include:

- Water flooding.
- Highly heterogeneous reservoirs.

Since FrontSim can be run on large models is also a good tool for:

- Screening of large geological models.
- Validation of up-scaled models.

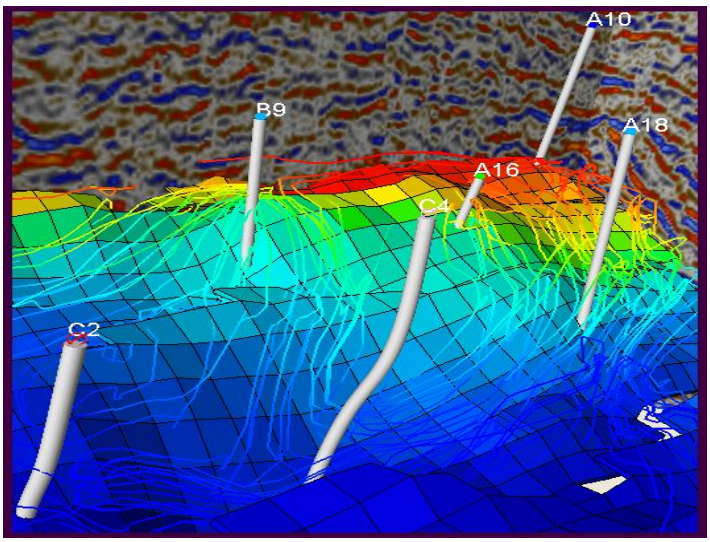

Fig.21.1: Streamline Simulator for some wells

Streamline Calculation

Given the fluid pressure in each of the cells, we can find the streamlines by selecting a number of so called starting points in the reservoir. Beginning at these starting points, we find the lowest pressure (or alternatively the highest) in close vicinity of the starting point, such as the steepest downhill we can find, and follow this downhill to the next point, etc. This process will eventually lead us to a producing well. When we have gone through this process for all the starting points, the reservoir should be covered by streamlines.

Streamline Representation

Since the streamlines describe the direction of flow at the moment in time that they are computed, they can be used as a computational tool to move the fluids along. The streamlines can be seen as a set of tubes representing the total reservoir volume and through which the fluids are moving. The tubes have exactly the same geological properties as the underlying geological model has. This approach divides the fluid flow into a set of 1-dimensional models and therefore reduces the problem to a computational simplicity that can have significant advantages. Since each streamline carries about the same volume, the density of the streamlines is an indication of the velocity of the fluid.

Streamline Simulation Advantages

The sequential process described above (solve pressure – move fluids along streamlines) constitutes one time step. An advantage with this approach is that the time steps can be very long and therefore the computational effort needed can be reduced.

The obvious advantage of a streamline method is the possibility it offers to do relatively quick simulations on large geologically and architecturally complex models. This is the main reason that the streamline method is a popular tool for validating up-scaling methods, measuring uncertainty, screening and ranking different model scenarios. Another advantage of the streamline method is the visual information it gives about the fluid flow pattern and connectivity in the reservoir. This can be used to study the effects of different well patterns on the flow, and to monitor which volumes, wells and boundaries that support a specific well at any given time.

Streamline Time Dependency

Even though a streamline simulation can be significantly faster than standard simulations, it can still be a time consuming affair if we want to model in detail all physical properties available in FrontSim. Some of the most important time dependencies are discussed below.

Number of Cells

The size of the model represented by the number of cells, will impact the time it will take to compute the pressure and move the fluids along. For example, for a model with 1 million grid cells it might take up to several minutes to compute the pressure and move the fluids one time step, depending somewhat on the complexity of the fluid description and the computational speed of the processor of your PC. Also, the memory requirement might be in the order of 1 gigabyte (Gb).

Fluid Description

The fluid description will impact the CPU time. The fastest model will be a low compressible 2-phase model (oil/water). The low compressibility will cause the pressure solution to be easier to solve and the requirement for time steps for accuracy is low. In addition, if the mobility and densities of the fluids are close, one time step might still result in a reasonably accurate (also from engineering perspective, that is, standard simulation) simulation result. For a one million-grid cell model, one time step might require 3 minutes of computer time with a state-of-the-art PC.

Time Steps

The number of time steps and the length of each time step will influence both the result and the time it will take to achieve a result. As we introduce more detail in the fluid description, we also potentially introduce a need to use more and shorter time steps. Such a factor is gravity segregation. To be accurate, more time steps are required even though FrontSim will include the effect in a onetime step solution.

Compressibility

Another factor is the compressibility introduced when using a 3-phase oil/water/gas model. To achieve close to engineering accuracy, we will have to enable the gravity segregation and use many time steps. In addition, the non-linearity of the model will cause more computational effort per time step. Note that FrontSim's streamline concept will allow the user to use only one long time step (possibly years), even for this type of model, but the engineering accuracy defined by standard simulation will be degraded.

The resulting set of partial differential equations cannot be solved by any analytical means due to its typical complexity in geometry, rock property and fluid description. Instead, a so called numerical approximation is used. Many types of numerical methods are available to solve these equations. Most use some form of a finite difference/volume method that divides the geometry into many small subsections called cells containing rock and fluid properties. These cells cover the whole domain and the fluid flow is represented as relationships between these cells. These relationships result in an equation system with a number of unknowns on the order of number of cells or higher. This equation system will have to be solved for every time step during the simulation. These methods might require very many small time steps, and because of this, a reservoir simulation process can be a very time consuming affair depending on the geological detail represented and fluid property modeled.