Analysis of the Performance of Streamline Simulation

Célio Maschio  
Departamento de Engenharia de Petróleo – Faculdade de Engenharia Mecânica  
UNICAMP Caixa Postal 6122 – Cep 13083-970  
Campinas – SP – Brasil  
celio@dep.fem.unicamp.br

Denis José Schiozer  
Departamento de Engenharia de Petróleo – Faculdade de Engenharia Mecânica  
UNICAMP Caixa Postal 6122 – Cep 13083-970  
Campinas – SP – Brasil  
denis@dep.fem.unicamp.br

1. Introduction

Streamline-based flow simulation has made significant advances in the last ten years. Some simulators already are fully 3D, account for gravity, fluid mobility change effects as well as moderated complex well controls. Most recent advances also allow for compressible flow and compositional displacements. A number of recent publications demonstrate how streamline-based simulation can be applied to several situations where traditional simulation, based on finite difference, have some limitations such as for very large models.

Streamline simulation method solves a three-dimensional problem by decoupling it into a series of one-dimensional problems and uses an IMPES numerical technique. The strategy consists on solving the pressure equation implicitly to compute a set of streamlines that represent flow in the reservoir. Each streamline represents a volumetric rate and acts as one-dimensional grid running perpendicular to the pressure contour in the reservoir. In this type of simulation, fluids move along streamlines, in contrast to conventional simulators, by which fluids are confined to the grid cells and move in an orthogonal direction to grid cell faces. As in a finite-difference IMPES approach, in a streamline simulator pressure is remained constant during a given time step and the streamlines are updated for each time step.

A common simplification among the streamline simulators is that capillary pressure is neglected. The main reason is that capillarity is a diffuse phenomenon. Capillary pressure is both significant along and across streamlines and therefore does not naturally lend itself to streamline simulation.

For a fluid/rock incompressible system, streamlines acts as connections between injectors (source) and producers (sink). Once the pressure field is computed (as in conventional form), the saturation distribution on the grid is mapped onto streamlines. For compressible systems, streamlines do not necessary go from injectors to producers. A grid block can acts as a source due to the rock compression or as a sink due to the phases expansion. This is more onerous in terms of number of streamlines to be traced. Another aspect is that for compressible problems, a more frequent pressure field updating is necessary, constraining to smallest values the maximum time step size between two consecutive pressure field solutions.

For incompressible systems, time step size can be orders of magnitude larger than the time step size in conventional simulators. This is a result of the elimination of the global grid CFL condition by decoupling fluid movement from underlying grid.

More numerical and mathematical details of streamline method can be found in Baker at al. (2001), Batycky (1997), Samier at al. (2001).
The main objective of this work is to evaluate the performance of streamline simulation and establish some insights about its applicability for some particular cases, analyzing the limitations and the situations in which it is or not suitable. First, a streamline simulator is applied for synthetic fields. The complexity of the problems is increased gradually, to evaluate the influence of several parameters and field characteristics such as: compressibility, heterogeneities and grid size. The effect of numerical parameters such as time step size is also verified. The petrophysical properties of SPE 10th Comparative Solution (Model 2) are used in the applications. These properties are highly heterogeneous and offer a good way to test this effect in the performance of the streamline simulator. This case is named in this work as “Field A”.

Streamline simulator is also applied to a Brazilian offshore field (“Field B”). The field has a large production history (about 30 years), 60 wells and an opening schedule of approximately 5 years. Time step size is gradually increased, in the period in which all wells are already opened, and the precision and simulation time relationship is evaluated. Finally, to benchmark the performance of the streamline simulator, a comparison to a traditional simulator is made.

2. Formulation

Streamline simulation uses an IMPES formulation for solution of the flow equations. Pressure field is solved implicitly while oil/gas/water phases saturations are solved explicitly. The basic governing equations are Darcy’s law and mass conservation. Considering a multi-phase incompressible flow, without capillary pressure effects, the Darcy’s law can be written in general coordinates as:

$$\ddot{u} = -\frac{k_{rj}}{\mu_j} \left( \nabla P_j + \rho_j \bar{g} D \right),$$

where $D$ is a reference depth below datum, $\bar{g}$ is gravitational acceleration, $k$ is permeability tensor, $k_{rj}$, $\mu_j$ and $\rho_j$ are relative permeability, viscosity and density of the phase $j$, respectively.

Conservation mass equation can be written as:

$$\nabla \cdot \ddot{u} = 0,$$

Combining equations (1) and (2), pressure field is calculated, and using equation 1, velocity filed is also calculated.

Material balance (saturation equation) is written as:

$$\Phi \frac{\partial S_j}{\partial t} + \ddot{u}_t \cdot \nabla f_j + \nabla \cdot \tilde{G}_j = 0$$

where $\ddot{u}_t$ is the total velocity for a given phase, $f_j$ is the fractional flow of phase $j$ and $\tilde{G}_j$ is the gravity component of fractional flow of phase $j$. Fractional flow is given by:

$$f_j = \frac{k_{rj}}{\mu_j} \left( \sum_{j=1}^{n_p} \frac{k_{rj}}{\mu_j} \right)^{-1}$$
For oil-water system, for example, \( f_j = (k_{rw} / \mu_{rw})/(k_{rw} / \mu_{rw} + k_{ro} / \mu_{ro}) \).

Gravity component of fractional flow is given by:

\[
\vec{G}_j = k_g f_j \nabla D \sum_{i=1}^{n_p} k_{ij} (\rho_i - \rho_j) / \mu_j
\]  
(5)

2.1 Time-of-flight

The time-of-flight is the fundamental definition of streamline simulation. It is used to a coordinate transformation from physical (3D) space to a one-temporal dimension (the time-of-flight dimension), as the fluid transport equations along streamlines are decoupled from underlying grid. The time-of-flight, \( \tau(s) \), is the time required by a given fluid mass (or fluid particle) to reach a distance, \( s \), along a streamline based on the velocity filed along the streamline, and is mathematically defined as:

\[
\tau(s) = \int_0^s \frac{\phi(\delta)}{\bar{u}_i(\delta)} d(\delta)
\]  
(6)

where \( \delta \) is a coordinate along streamline and \( d() \) is infinitesimal time for a particle goes from node 1 to node 2, as in figure 1

![Figure 1 - Streamline and time-of-flight](image)

Equation 6 leads to write:

\[
\left| \bar{u}_i \right| \frac{\partial}{\partial \delta} \equiv \bar{u}_i \cdot \nabla = \phi \frac{\partial}{\partial \tau}
\]  
(7)

The identity and equation 7, combined to equation 3 allows writing saturation equation in one dimension:

\[
\frac{\partial S_j}{\partial t} + \frac{\partial f_j}{\partial \tau} + \frac{1}{\phi} \nabla \cdot \vec{G}_j = 0
\]  
(8)

Equation 8 is known as pseudo-1D saturation equation, because the gravity term is not aligned (on seldom aligned) along a streamline direction. Therefore, equation is solved by an operator-splitting that simply consists in solving the equation in two steps: the first is a “convective step” which is taken along the streamlines; the second is a “gravity step”, in which phases are segregated vertically according to the phase densities.
The convective step is written as:

$$\frac{\partial S_j^c}{\partial t} + \frac{\partial f_j}{\partial \tau} = 0$$

(9)

and the gravity step is written as:

$$\frac{\partial S_j^g}{\partial \tau} + \frac{1}{\phi} \nabla \cdot G_j = 0$$

(10)

2.1 Time-of-flight and compressible system

For compressible system, the time of-flight can be written in a similar way as previously described, by defining a diffusivity term, $\alpha$, as follow:

$$\alpha = \frac{k}{\phi(s) \mu c_t}$$

(11)

being $c_t$ the total compressibility of the system.

This leads to write the time-of-flight for a compressible system as:

$$\tau(s) = \int_0^s \frac{d(\delta)}{\sqrt{\alpha}}$$

(12)

that is called diffusive time-of-flight (Datta-Gupta, 2001).

To summarize, the main steps and ideas of streamline simulation is:

(i) tracing streamlines based on a velocity field solved implicitly from basic flow equations using a finite difference or a finite element technique;

(ii) computing travel time (or time-of-flight) along streamlines;

(iii) decoupling the transport equations (concentration and saturation equations) using a coordinate transformation from physical space to the time-of-flight coordinates following flow directions;

(iv) solving the transport equations along streamlines;

(v) updating streamlines to account for compressibility, gravity, mobility effects and change field conditions (new wells shut in, for example).

3. Methodology

3.1 Descriptions of the tested models

Field A: SPE 10th comparative solution project (SPE 10)

The model dimensions are $1200 \times 2200 \times 170$ (ft), with a grid size of $60 \times 220 \times 85$ and the fine scale cell size is $20 \times 10 \times 2$ ft. The $x$ direction permeability is equal to $y$ direction, varying from 0.5 to 20000 mD. $K_z$ is different from $x$ and $y$ and varies from 0.5 to 6000 mD. A porosity map is also used, varying from practically 0 to 0.50. Four producer wells are used, each one in the corners of the models and an injector well in the center. More details for this model can be found in Christie and Blunt (2001).
Firstly, heterogeneous petrophysical properties from 10th comparative solution project were replaced by homogeneous properties (permeability and porosity). In Table 1 are presented the size of homogeneous models.

Table 1 - Homogenous models

<table>
<thead>
<tr>
<th>Case SPE10 homogeneous</th>
<th>Model</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
</tr>
</thead>
<tbody>
<tr>
<td>Grid</td>
<td></td>
<td>30x110x42</td>
<td>20x74x28</td>
<td>15x55x21</td>
<td>12x44x17</td>
<td>10x37x14</td>
</tr>
<tr>
<td>Blocks</td>
<td></td>
<td>138600</td>
<td>41440</td>
<td>17325</td>
<td>8976</td>
<td>5180</td>
</tr>
</tbody>
</table>

Four heterogeneous models were generated using a geometrical average technique for upscale absolute permeability from fine grid and an arithmetic average for upscale porosity. In Table 2 are presented the grid size of heterogeneous models.

Table 2 - Heterogeneous models from 10th comparative project

<table>
<thead>
<tr>
<th>Case SPE10 heterogeneous</th>
<th>Model</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
</tr>
</thead>
<tbody>
<tr>
<td>Grid</td>
<td></td>
<td>10x37x14</td>
<td>12x44x17</td>
<td>15x55x21</td>
<td>17x63x24</td>
</tr>
<tr>
<td>Blocks</td>
<td></td>
<td>5180</td>
<td>8976</td>
<td>17325</td>
<td>25704</td>
</tr>
</tbody>
</table>

Field B: Offshore Filed of Campos Basin, Brazil

Field B is an offshore field of Campos Basin, Brazil. The reservoir is formed by sandstone turbidites confined by faults with good porosity and permeability. The STOIP of the field is approximately 100 MMm$^3$ and the main production mechanism is solution gas drive. It is a developed field, with more than 20 years of production. The drainage is accomplished through 33 oil production wells and 13 water injector wells. The available data before production started counting with nine seismic lines, 8 perforated wells, 4 oil analyses (PVT), formation test analysis, interpreted electric logs and data cores of 3 wells. The data cores comprise porosity and permeability sample, 7 analyses of relative permeability, 10 capillary pressures and rock compressibility. Datum depth is located at 3000 m, water oil contact is located at 3100 m and gas oil contact is located at 2900 m.

4. Results and discussions

4.1 Field A

Figures 2 to 4 show a sensitivity analysis of the streamline simulator referent to the maximum time step size for the five homogeneous models (in Table 1). The strategy consisted in the variation of maximum time step size for streamline and for Eclipse the default maximum time step size (365 days) was maintained. For cumulative oil production, it can be seen a slightly overestimation for streamline simulator, when compared to the Eclipse, for all models, but deviation due to time step size is little. More pronounced deviations occur on water production (Fig. 3). It is evident that for time step equal to 550 days water cut curve deviates significantly of the curve obtained with Eclipse. It can also be seen, still for water production, that for grid size variation, the curves exhibit similar trend for each time step. With respect to reservoir pressure, Fig. 4 shows the curves for time step equals to 100 days (Fig. 4a) and 550 days (Fig. 4b). Practically, all curves point out a higher pressure drop referent to streamline simulator and the deviations depend on the grid size. The same behavior is observed for time step 365 days.
Comparison of CPU time of the two simulation techniques is presented in Fig. 5. For maximum time step size equal to 100 days, fixed for streamline simulator, CPU time for the Eclipse is smallest. For time steps of 365 and 550, streamline is faster than Eclipse. This shows that for this homogenous reservoir, the speedup of streamline simulator is not very significant.

Figure 2 - Comparison of cumulative oil rate for different time steps (Streamlines) with Eclipse (FD)

Figure 3 - Comparison of water cut for different time steps (Streamlines) with Eclipse (FD)

Figure 4 - Comparison of reservoir pressure for different time steps (Streamlines) with Eclipse (FD): (a) Time step = 100 days, (b) Time step = 550 days
Figures 6 to 9 present production parameters for the Eclipse and streamline simulation referent to the heterogeneous models. A maximum time step size of 100 days was fixed for streamline simulations. Firstly, it can be seen a very good agreement of the results for all models. The CPU time for heterogeneous models is shown in Fig. 10. It is evident, in this case, that there is an excellent speedup referent to the streamline simulator. For model 4, for example, Eclipse taken approximately 960 minutes, while streamline taken only 22 minutes, resulting in a speedup factor of about 43. Another important aspect from this result is that run time for Eclipse increases exponentially with the increase of number of blocks, while for streamline there is a linear increase in run time. This behavior can also be observed for homogenous models, but streamline can provide some advantages only for very large grid models.

Effect of compressibility on streamline simulation was also evaluated. Fig. 12 shows comparison of CPU time for compressible and incompressible models. There is a considerable increase (approximately a factor of 2.5) on run time for compressible models.
Figure 7 - Results from heterogeneous Model 2

Figure 8 - Results from heterogeneous Model 3

Figure 9 - Results from heterogeneous Model 4
4.1 Field B

Table 3 shows run time for FD and streamline simulation for Field B. Using a maximum time-step equals to 80 days for SL, the run time (4166 s) is greater than FD (3640 s). Only for time-step equals or greater than 169 days, SL run time is lower than FD. As can it be seen from Fig. 12 (a), for time-step free in SL, there is more deviate in oil rate prediction. Reservoir pressure is slightly affected by time-step size in SL, but results compared to FD are not very in agreement.

Field B is much more complex in terms of fluid properties and reservoir conditions (large production history and well opening schedule). Water-oil contact and gas-oil contact are present with capillary pressure. Efficiency and mainly run time of streamline simulation is strongly dependent on the fluids properties and on the frequency of changes in the well conditions or the number of new wells included gradually.

According to these results, it is possible to observe that depending on the type of application, where a great accuracy is not necessary, streamline can be applied. For instance, risk analysis or general production strategy definition. In other cases where accurate results are necessary, there is a limitation inherent to streamline simulation, especially when complex full-physics parameters are present. This limitation can be associated either to run time and/or to reproduction (accuracy) of results.

<table>
<thead>
<tr>
<th>FD run time (s)</th>
<th>SL run time (s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>3640</td>
<td>TS&lt;sub&gt;max&lt;/sub&gt; = 80</td>
</tr>
<tr>
<td></td>
<td>4166</td>
</tr>
</tbody>
</table>

Figure 10 - Run time for Eclipse and Streamline for heterogeneous models

Figure 11 - Effect of compressibility on the run time for Streamline
5. Conclusions

Streamline simulation is very efficient, in terms of run time and accuracy, for incompressible or slightly compressible models, with high degree of heterogeneities and large grid size, which represent a limitation for traditional finite difference simulators. For this cases streamline simulation have clear advantages compared to traditional simulators.

Once streamline simulation must necessarily neglect or simplify some important physics in order to run quickly, it should not be intended to match or rigorously predict field recovery when complex physics related to fluids properties and fluids flow must be considered. The main limitation is when highly compressible fluids are present in the model. In these situations, conventional simulation is required for more rigorous and reliable evaluations.

The knowledge of the limitations and the combination of both tools seems to be very interesting for many reservoir engineering applications.

6. References


