A 3-D EDGE-BASED HIGHER ORDER FINITE VOLUME PROCEDURE FOR THE SIMULATION OF TWO PHASE FLOW OF OIL AND WATER IN POROUS MEDIA USING A MODIFIED IMPES APPROACH

Darlan Karlo. Elisiário. de Carvalho, dkarlo@uol.com.br
Rogério Soares da Silva, rogsoares@yahoo.com.br
Paulo Roberto Maciel Lyra, prmlyra@ufpe.br
Ramiro Brito Willmersdorf, ramiro@willmersdorf.net
Departamento de Engenharia Mecânica, UFPE, Av. Acadêmico Hélio Ramos, s/n, CEP: 50740-530, Recife - PE – Brazil.

Abstract. In the present paper we present a higher order finite volume method with a "Modified Implicit Pressure, Explicit Saturation" (MIMPES) to model the 3-D incompressible and immiscible two-phase flow of water and oil in heterogeneous and anisotropic porous media. We use the vertex centered finite volume method with an edge-based data structure to discretize both, the elliptic pressure and the hyperbolic saturation equations. The classical IMPES approach involves three steps: first, the pressure equation is solved implicitly from an initial saturation distribution; then, the velocity field is computed explicitly from the pressure field and finally the saturation equation solved explicitly. This saturation field is then used to re-compute the pressure field and the process follows until the end of the simulation is reached. Due to explicit solution of the saturation equation, severe time restrictions are imposed on the simulation. In order to circumvent this problem, an edge-based implementation of the MIMPES method was used. In the modified IMPES method used here, pressure equation is solved and the velocity field is computed much less frequently than the saturation field, using the fact that, usually, the velocity field varies slowly throughout the simulation implying that the saturation field can be updated several times before we have to update the pressure/velocity fields. Following the work of Hurtado, we have used the mean relative variation of the velocity field throughout the simulation to automatically control the updating process, allowing for much larger time-steps in a very simple way. The solution of the pressure equation is performed using a modification of Crumpton's two step approach which was designed to handle material discontinuity properly. In the first step, gradients (fluxes) are computed. In the second step, these fluxes are used to compute the diffusion terms. The saturation equation is solved explicitly using an edge-based implementation of a modified second order MUSCL (Monotone Upwind Scheme for Conservation Laws) type method. Some examples are presented in order to validate the proposed formulation. Results match quite well with others found in literature.

Keywords*: Porous media, MIMPES, edge data structure, finite volume method, high order approximation.*

1. INTRODUCTION

One of the most popular methodologies used to describe the fluid flow in petroleum reservoir simulation is the IMPES (Implicit Pressure Explicit Saturation) procedure (Ewing, 1983; Chen et al., 2002; Hurtado et al., 2006; Carvalho et al. 2007). In this technique, a sequential time stepping procedure is used to split the computation of the pressure field from the saturation field. In the classical IMPES method, initially, the pressure equation is solved implicitly from an initial saturation distribution, and then, velocities are computed from this pressure field. In sequence, the velocity field is used as an input for the saturation equation, which is finally solved explicitly. The process is repeated until the end of the analysis.

The pressure field is governed by a parabolic/elliptic type equation that can have strong discontinuous coefficients (i.e. permeabilities) and, in general, the saturation equation is similar to a convection-diffusion type equation, in which the diffusion coefficients are associated to capillary effects. In many situations, capillarity is small and can be neglected. In such cases, the saturation equation behaves essentially as a first order non-linear hyperbolic conservation law.

The pressure field calculation involves the solution of a system of equations that represents a higher CPU cost when compared to the saturation equation evaluation. For large scale problems CPU cost can become prohibitive which leads researchers to find other ways to make simulation viable.

Due to explicit solution of the saturation equation, severe time restrictions are imposed on the simulation. In order to circumvent this problem, an edge-based implementation of the modified IMPES method was used. In the Modified IMPES method (MIMPES) used here, pressure equation is solved and the velocity field is computed much less frequently than the saturation field, using the fact that, generally, the velocity field varies slowly throughout the simulation implying that the saturation field can be updated several times before we have to update the pressure/velocity fields (Chen et al., 2002; Hurtado et al., 2005; Hurtado et al., 2006).

The governing equations are discretized by a node-centered conservative fully finite volume formulation with an edge-based data structure (Carvalho et al., 2005; Carvalho et at., 2007), where geometrical coefficients, computed once in a pre-processing stage, are associated to the edges and nodes of the primal mesh to define the surface and the volume of all control volumes, respectively. This formulation has been chosen due to the fact that node-centered FV schemes are usually superior to cell centered schemes in terms of memory usage and because edge-based data structures are known to be more computationally efficient than their element-based counterparts (Luo et al., 1995; Lyra and Morgan, 2002; Rees et al., 2004).

In our approach, the finite volume method is used to accurately solve both, the elliptic pressure equation and the hyperbolic saturation equation. For the solution of the pressure equation, we devise an alternative way of computing continuous diffusive fluxes through control surfaces faces in heterogeneous media with strong discontinuous coefficients by a modification of Crumpton's two step approach (Crumpton et al., 1997). The hyperbolic saturation equation is solved using a higher order MUSCL type scheme adapted for use with multidimensional unstructured meshes (Lyra and Morgan, 2002; Carvalho et al., 2007), with the slope limiter proposed by Woodfield et al. (2005), which has some resemble with the multidimensional slope limiter of Zalezak (1979).

2. MATHEMATICAL FORMULATION

In this section, we briefly describe the governing equations for incompressible, immiscible two-phase flows of water and oil through rigid porous media. This model (which can be directly extended to miscible, three phase flow) is obtained combining Darcy's Law with the mass conservation equation for each phase. The model adopted here has been successfully used by many authors (Ewing, 1983; Chen et al., 2002; Hurtado et al., 2006; Carvalho et al., 2005; Carvalho et al., 2007), though it is still not commonly used in commercial reservoir simulators.

First, we assume that the phase velocities obey the Darcy's law, which, ignoring gravitational effects can be written for phase *i*, as

$$\vec{v}_i = -\underset{\sim}{\lambda}_i \nabla P_i \tag{1}$$

where the phase mobility tensor $\underset{\sim}{\lambda}_i$ is defined as

$$\underset{\sim}{\lambda}_i = \underset{\sim}{K}\frac{k_i}{\mu_i} \text{ or } \underset{\sim}{\lambda}_i = \underset{\sim}{K}\lambda_i \tag{2}$$

In Equation (2), $\underset{\sim}{K}$ denotes the absolute permeability tensor of the rock, $\lambda_i = k_i/\mu_i$ is the scalar phase mobility, with k_i being the phase relative permeability and μ_i the phase viscosity. Henceforth, we will assume incompressible medium and fluids. We will also ignore the capillary pressure and assume that $P = P_w = P_o$, where (w) and (o) stand, respectively, for the wetting (water) and the non-wetting (oil) phases. Additionally, conservation of mass for each phase *i* can be written as

$$-\nabla\cdot\left(\rho_i\vec{v}_i\right) + q_i = \frac{\partial\left(\phi\rho_i S_i\right)}{\partial t} \tag{3}$$

In Equation (3), ϕ is the porosity, i.e. fraction of the rock which can be occupied by fluids, q_i denotes sources or sinks, ρ_i is the phase density and S_i is the saturation of phase *i*, which represents the percentage of the available pore space occupied by this phase. Due to this last definition, we can write

$$S_o + S_w = 1 \tag{4}$$

Combining equations (1) to (4) and after some algebraic manipulation we obtain the following pressure equation

$$\nabla\cdot\left(\underset{\sim}{\lambda}\nabla P\right) = -Q \text{ or } \nabla\cdot\vec{v} = Q \tag{5}$$

where, $\underset{\sim}{\lambda} = \underset{\sim}{\lambda}_o + \underset{\sim}{\lambda}_w$ is the total fluid mobility tensor, $\vec{v} = \vec{v}_o + \vec{v}_w = -\underset{\sim}{\lambda}\nabla P$ is the total velocity field and $Q = Q_w + Q_o$, with $Q_i = \left(q_i/\rho_i\right)$, is the total injection or production specific rate. By introducing the fractional flow function $f_i = \lambda_i/\left(\lambda_o + \lambda_w\right)$, we can also derive a hyperbolic equation for the water saturation, which can be written as

$$\phi\frac{\partial S_w}{\partial t} + \nabla\cdot\vec{F}_w(S_w) = Q_w \tag{6}$$

The term $\vec{F}_w = f_w \vec{v}$ is the flux function which is dependent on the water phase saturation. As it can be seen, the pressure and saturation fields are connected through the total velocity $\vec{v}$.

3. FINITE VOLUME FORMULATION

In order to discretize Equations (5) and (6), i.e. the pressure and the saturation equations, respectively, we have adopted a node centered, median dual full finite volume (FV) technique, in which the coefficients necessary to our calculation are associated to the edges and to the nodes of the mesh (Luo et al., 1995; Lyra et al., 2004; Rees et al., 2004; Carvalho et al., 2007). These edge and node coefficients are pre-computed in a pre-processing stage from the more traditional element data structure which is commonly used in the finite element method (Smith, 2001).

Even though, there is, in principle, no restriction to the shape of the elements utilized to discretize the spatial domain, it is important to keep in mind that FV edge-based schemes are only linearly preserving (i.e., they exactly represent a linear field) on triangular (2D), tetrahedral (3D) or structured quadrilateral (2D) and hexahedral (3D) meshes (Müller, 1998), even though linearity can be recovered through the use of "ghost edges". Therefore, extra care must be taken when using different element types, especially when considering highly distorted meshes.

The median dual control volumes (CV) adopted are built connecting centroids of elements to the middle point of the edges that surround a specific mesh node, even though different control volumes could be used. In edge-based node centered schemes, fluxes are usually integrated on the dual mesh through one or more loops over the edges, and the computational cost is, essentially, proportional to the number of edges of the mesh. In order to properly handle material discontinuities we perform the integration over the whole domain in a sub-domain by sub-domain approach, where a sub-domain is defined by a group of elements that share the same physical properties such as permeability and porosity.

Node centered finite volume schemes in which physical properties are uniform in the elements of the primal mesh are named "cell distributed schemes" in opposition to the so called "point distributed schemes" in which properties are uniform inside the control volumes of the dual mesh (Verma 1996).

In the present work, we have chosen to use the cell distributed methodology due to the easiness of associating rock properties to sub-domains which encompass groups of elements (e.g. triangles or tetrahedral) that naturally fit to reservoir bed boundaries.

A detailed description of all steps to obtain the discretized pressure, velocity and saturation equations are detailed in Carvalho et al. (2007). Here, these steps were omitted and only the final equations are presented.

3.1. Implicit pressure equation

For 2-D and 3-D problems, Eq. (7) is the discretized form of the pressure equation which is solved through a system of linear equations assembled edge-by-edge.

$$\sum_{R=1}^{Ndom}\left[\sum_{L_{I(\Omega_R)}}\left(-\underset{\sim}{\lambda}_{IJ_L}^{\Omega_R}\left(\frac{\left(\nabla P_I^{\Omega_R}+\nabla P_{J_L}^{\Omega_R}\right)}{2}-\left(\frac{\left(\nabla P_I^{\Omega_R}+\nabla P_{J_L}^{\Omega_R}\right)}{2}\cdot\vec{L}_{IJ_L}\right)\vec{L}_{IJ_L}+\frac{\left(P_{J_L}-P_I\right)}{\left|\Delta_{IJ_L}\right|}\vec{L}_{IJ_L}\right)\right)\cdot\vec{C}_{IJ_L}^{\Omega_R}\right] = Q_I V_I - \sum_{R=1}^{Ndom}\left(\sum_{L_{I(\Gamma_R)}}\vec{v}_{IJ_L}^{\Gamma}\cdot\vec{D}_{IJ_L}^{\Omega_R}\right) \tag{7}$$

The nodal gradients presented in Eq. (7) can be approximated for 2-D domains by Eq. (8), as

$$\nabla P_I^{\Omega_R} = \frac{1}{V_I^{\Omega_R}}\left[\sum_{L_{I(\Omega_R)}}\vec{C}_{IJ_L}^{\Omega_R}\frac{\left(P_I+P_{J_L}\right)}{2}+\sum_{L_{I(\Gamma_{RE})}}\vec{D}_{IJ_L}^{\Omega_R}\frac{\left(5P_I+P_{J_L}\right)}{6}+\sum_{L_{I(\Gamma_{RI})}}\vec{D}_{IJ_L}^{\Omega_R}\frac{\left(5P_I+P_{J_L}\right)}{6}\right] \tag{8}$$

For 3-D domains gradients can be computed as

$$\nabla P_I^{\Omega_r} = \frac{1}{V_I^{\Omega_r}}\left[\sum_{J_L=1}^{NN^{(\Omega_r)}}\frac{\left(P_I+P_{J_L}\right)}{2}\vec{C}_{IJ_L}^{\Omega_r}+\sum_{J_L=1}^{NN^{(\Gamma_{RE})}}\frac{\left(6P_I+P_{J_L}+P_{J_H}\right)}{8}\vec{D}_{IJ_LJ_H}^{\Omega_r}+\sum_{J_L=1}^{NN^{(\Gamma_{RI})}}\frac{\left(6P_I+P_{J_L}+P_{J_H}\right)}{8}\vec{D}_{IJ_LJ_H}^{\Omega_r}\right] \tag{9}$$

Note that, even though Equation (9) can be assembled in a pure edge-based manner, we have decided to use an auxiliary data structure for the boundary faces (external and between domains) as there is no clear improvement by using a pure edge-based data structure in this case.

In this work, all calculations are performed using an parallel reservoir simulator (Silva, 2008) written in C++ in which we have incorporated some free packages, such as the FMDB (Seol, 2005), used for mesh management, the PARMETIS, used for mesh partioning and the PETSc used as a library of linear solvers and pre-conditioners. Due the fact that the discretization of the pressure equation generates a sparse, non-symmetric and positive defined, global matrix, the Generalized Minimum Residual Solver (GMRES).

After pressure field is computed, nodal gradients are explicit recovered by Eq. (8), in 2-D, or Eq. (9), in 3-D, and the mid-edge velocity field is obtained by Eq. (10)

$$\vec{v}_{IJ_L}^{\Omega_R *} = -\underset{\sim}{\lambda}_{IJ_L}^{\Omega_R} \left(\frac{\left(\nabla P_I^{\Omega_R} + \nabla P_{J_L}^{\Omega_R}\right)}{2} - \left(\frac{\left(\nabla P_I^{\Omega_R} + \nabla P_{J_L}^{\Omega_R}\right)}{2} \cdot \vec{L}_{IJ_L} \right) \vec{L}_{IJ_L} + \frac{\left(P_{J_L} - P_I\right)}{\left|\Delta_{IJ_L}\right|} \vec{L}_{IJ_L} \right) \tag{10}$$

In this work, we have dealt only with isotropic porous media, even though there is no restriction to handle anisotropic media (Carvalho et al., 2009), therefore we have assumed that $\underset{\sim}{\lambda}_{IJ_L}^{\Omega_R} = \underset{\sim}{K}^{\Omega_R} \lambda_{IJ_L}$, where $\underset{\sim}{K}_{IJ_L}^{\Omega_R} = k^{\Omega_R} \underset{\sim}{I}$, with $k^{\Omega_R} = \text{constant}$ for each sub-domain Ω_R, and $\underset{\sim}{I}$ is the identity matrix. The edge values of the scalar mobility terms are approximated using a mid-point rule in order to formally guarantee second order accuracy (Rees, 2004), i.e. $\lambda_{IJ_L} = \left(\lambda_I + \lambda_{J_L}\right)/2$ and viscosity is constant under the assumption of incompressible flow.

Now, we can redefine Eq. (7) using this new mid-edge velocity approximation as

$$\sum_{R=1}^{Ndom} \left(\sum_{L_{I(\Omega_R)}} \vec{v}_{IJ_L}^{\Omega_R *} \cdot \vec{C}_{IJ_L}^{\Omega_R} + \sum_{L_{I(\Gamma_R)}} \vec{v}_{IJ_L}^{\Gamma} \cdot \vec{D}_{IJ_L}^{\Omega_R} \right) = Q_I V_I \tag{11}$$

and *Ndom* refers to the number of domains that surrounds node *I*.

3.2. The explicit saturation equation

Usually, in petroleum reservoir simulators, the discretization of the advective term that characterize the hyperbolic saturation equation is performed by the classical first order upwind type method, which is capable of completely eliminating spurious oscillations at the cost of introducing a large amount of artificial diffusion, (Ewing, 1983). On the other hand, pure second order schemes produce physically unrealistic results, with overshoots and/or undershoot in the vicinity of sudden changes in the saturation field (i.e. shocks).

In the present paper, we have used an edge-based higher order upwind type method which was developed by Woodfield et al. (2005). In our preliminary tests, this scheme has proved to be more robust than other edge-based schemes that rely only on gradient extrapolation or artificial dissipation schemes such as those presented in Lyra and Morgan (2002) and Carvalho et al. (2005). The method proposed by Woodfield et al. (2005), is essentially a variation of the MUSCL (Monotone Upstream Scheme for Conservation Laws) scheme of Van Leer (Hirsch, 1988) in which local boundness is reinforced by using a limiting procedure that resembles the Zalesak's procedure (Zalesak, 1979). In this case monotonicity is enforced through a scalar limiting function applied to the piecewise polynomial reconstruction procedure. The monotonic constrains are normally introduced in the form of slope limiters in order to avoid undershoots and overshoots in the numerical solution. Below, we present a brief description of the scheme. For further details, see Woodfield et al. (2005).

By integrating Eq. (6) and applying the divergence theorem we can write,

$$\int_{\Omega} \phi \frac{\partial S_w}{\partial t} \partial\Omega + \int_{\Gamma} \vec{F}_w\left(S_w\right) \cdot \vec{n}\, \partial\Gamma = \int_{\Omega} Q_w \partial\Omega \tag{12}$$

The source term, which was treated using a simple fractional step approach (Le Veque, 1992), is non zero only at production wells and for a particular mesh node *I*, the second term in the left hand side is approximated as,

$$\int_{\Gamma_I} \vec{F}_w\left(S_w\right) \cdot \vec{n}\, \partial\Gamma_I \cong \sum_{L_I} \vec{F}_{IJ_L(w)} \cdot \vec{C}_{IJ_L} = \sum_{L_I} \frac{1}{2} \left[\left(\vec{F}_I^{-}\left(S_{I(w)}^{-}\right) + \vec{F}_{J_L}^{+}\left(S_{J_L(w)}^{+}\right) \right) \cdot \vec{C}_{IJ_L} - \alpha_{IJ_L}\left(S_{J_L(w)}^{-} - S_{I(w)}^{+}\right) \right] \tag{13}$$

where $\alpha_{IJ_L} = \left|\vec{v}_{IJ_L}\right| \left|\Delta f_{IJ_L(w)} / \Delta S_{IJ_L(w)}\right|$, with $\Delta f_{IJ_L(w)} / \Delta S_{IJ_L(w)} = \left(f_{J_L(w)} - f_{I(w)}\right) / \left(S_{J_L(w)} - S_{I(w)}\right)$ and the superscripts (-) and (+) are used to indicate that fluxes are computed using the following linear extrapolated saturation values,

$$S^{+}_{I(w)} = S_{I(w)} + \frac{\psi^{*}_{I}}{2}\left(\nabla S_{I(w)} \cdot \overrightarrow{IJ_L}\right) \text{ and } S^{-}_{J_L(w)} = S_{J_L(w)} + \frac{\psi^{*}_{J_L}}{2}\left(\nabla S_{J_L(w)} \cdot \overrightarrow{IJ_L}\right) \tag{14}$$

where $\overrightarrow{IJ_L}$ is the length vector in the edge direction (i.e. $\vec{x}_{J_L} - \vec{x}_I$), and ψ^{*}_{I} is a slope limiter which must smoothly switch from one (second order scheme) to zero (first order scheme) in the vicinity of saturation shocks. This slope limiter is computed using the following expression,

$$\psi^{*}_{I} = \psi_I \psi_{IJ_L} \tag{15}$$

In Equation (15), ψ_I is responsible for switching the scheme from second order to first order whenever necessary and ψ_{IJ_L} is responsible for the edge interpolative boundedness, i.e. it guarantees that the extrapolated values of the saturation values throughout the edge remain between $S_{I(w)}$ and $S_{J_L(w)}$.

Before defining ψ_I and for convenience of notation, we define the following dimensionless parameter,

$$\gamma_I = \frac{S_{I(w)} - S_{J_L\,min(w)}}{S_{J_L\,max(w)} - S_{J_L\,min(w)}} \tag{16}$$

In Equation (16), $S_{J_L\,min(w)}$ and $S_{J_L\,max(w)}$ are, respectively, the minimum and the maximum values of the saturation considering all nodes directly connected to node *I* (i.e. J_L) excluding *I* itself. In multi-dimensional problems, the saturation is said to be "bounded" for every node *I* in the computational domain, if the following relation is true,

$$0 \le \gamma_I \le 1 \tag{17}$$

The use of the second order scheme at the whole computational domain except where Eq. (17) applies is not a safe strategy because there is not a smooth transition between the second order and the first order schemes. A safer approach consists in introducing a user defined free parameter which ranges from $0 \le \delta \le 0.5$, such as,

$$\delta \le \gamma_I \le 1 - \delta \tag{18}$$

Using the definitions of equations (16) and (18), we can compute the parameter ψ_I as,

$$\begin{aligned}
&\psi_I = 1 && \text{if} \quad \left(S_{J_L\,\max(w)} - S_{J_L\,\min(w)}\right) \le 10^{-20} \\
&\psi_I = 0 && \text{if} \quad \gamma_I \ge 1 \quad \text{or} \quad \gamma_I \le 0 \\
&\psi_I = 1 && \text{if} \quad \delta \le \gamma_I \le (1-\delta) \\
&\psi_I = \frac{\gamma_I}{\delta} && \text{if} \quad 0 < \gamma_I < \delta \\
&\psi_I = \frac{(1-\gamma_I)}{\delta} && \text{if} \quad (1-\delta) < \gamma_I < 1
\end{aligned} \tag{19}$$

It is worth mentioning that the first requirement avoids division by zero in the definition of γ_I. In the present paper, we have used $\delta = 0.2$. Note that, smaller values of this parameter will turn the scheme less diffusive as the limiter will tend to switch on and off more abruptly, as δ tends to zero, while larger values of the parameter will imply in a smaller range of values of γ for which the second order scheme is used.

On the other hand, to ensure edge interpolative boundedness the parameter ψ_{IJ_L} can be defined as,

$$\psi_{IJ_L} = \max(0, \theta), \text{ with } \theta = \min\left(1, \frac{2\left(S_{J_L(w)} - S_{I(w)}\right)}{\psi_I\left(\nabla S_{I(w)} \cdot \overrightarrow{IJ_L}\right)}\right) \tag{20}$$

As previously mentioned, we have chosen this method due to its robustness and relatively low computational cost. Whenever using elements with high aspect ratios, which are common in mesh adaptive processes, other alternatives, such as the gradient extrapolation approach or the artificial dissipation scheme ((Lyra and Morgan, 2002; Carvalho et al., 2005), have, respectively, produced erroneous solutions with noticeable over and undershoots or overly diffusive solutions (Carvalho et al, 2007). At least, in the case of the gradient extrapolation method, errors associated to the gradient recovery can be blamed for the lack of monotonicity of the final solution.

4. MODIFIED IMPES APPROACH (MIMPES)

The IMPES method is a segregated type method in which the flow equations are manipulated in order to produce an elliptic pressure equation, solved implicitly and a hyperbolic type saturation equation, which is then solved explicitly. In classical IMPES method the pressure and the saturation fields are updated simultaneously, i.e. the pressure field is solved assuming a time step which is also used for the velocity and the saturation fields. In fact, for incompressible fluid flow, the pressure equation is elliptic, and the time step used for advancing the solution of the system of equations as a whole is defined by the explicit solution of the hyperbolic saturation equation, which is limited by the CFL condition.

On the other hand, the Modified Implicit Pressure Explict Saturation approach (MIMPES) consists in assigning larger time steps to the implicit pressure equation than those used to solve the saturation equation (limited by the CFL restriction). The latter is solved repeatedly until the saturation filed reaches the same instant of the pressure field. The major advantage of this strategy is that the implicit pressure equation, which represents more than 90% of all calculations, is solved several times less than the explicit saturation equation.

The MIMPES algorithm, we are using in the present paper, is a slightly edge-based modification of the original element-based algorithm proposed by Hurtado et al. (2006). We have chosen this algorithm because it appears to give more robust results than others found in literature (Chen et al, 2004), in a simpler manner and at a lower computational cost. Besides it can be easily adapted for our edge-based data structure, which, essentially, makes it readily available for use in 2-D and 3-D simulations. The basic difference between the strategies proposed in Hurtado et al. (2006) and Chen et al. (2004) is related to the controlling of the time range used to update the pressure and the velocity fields. The time step controlling proposed by Chen is based on the maximum local saturation variation, while in the Hurtado's technique, the same task is based on the velocity field variation.

An algorithm containing the basic steps to implement the modified IMPES is presented in Fig. (4.1). After calculating the velocity field, a L2 norm, $\left|\Delta\vec{v}_T\right|^n$, of all mid-edge velocities is calculated. The new time-step (Δt_P^{n+1}) for the pressure equation is computed through Eq. (21), as

$$\Delta t_P^{n+1} = \frac{DVTOL}{\left|\Delta\vec{v}_T\right|^n}\Delta t_P^n \tag{21}$$

At the very first simulation time step, Δt_P^n assumes, the value defined by the CFL restriction, i.e. $\Delta t_P^{n=0} = \Delta t_{CFL}$. From a slightly modification on Hurtado's work, the velocity L^2 norm can be obtained by edges as

$$\left|\Delta\vec{v}_T\right|^n = \sqrt{\sum_{IJ_{\Omega_r}}\left(\frac{\left|\left(\vec{v}_{IJ}^{\Omega_r,n} - \vec{v}_{IJ}^{\Omega_r,n-1}\right)\cdot\vec{C}_{IJ}^{\Omega_r}\right|}{\left|\vec{C}_{IJ}^{\Omega_r}\right|}\right)^2} \tag{22}$$

```
Sw(t=0);
while (simulation_time < final_time){
    loop over domains {
        loop over edges and boundary face (if 3-D) to:
          pressure field calculation;
    }
    Implicit_ts = true;
    Explicit_ts = true;
    sum_time = 0;
    while (Explicit_ts){
        Loop over domains{
            If (Implicit_ts){
                Loop over edges and boundary faces (if 3-D) to:
                    Gradient recovery;
```

Loop over edges to:{
Velocity field calculation;

$$\left|\Delta\vec{v}_T\right|^n = \sqrt{\sum_{IJ_{\Omega_r}}\left(\frac{\left|\left(\vec{v}_{IJ}^{\,\Omega_r,n} - \vec{v}_{IJ}^{\,\Omega_r,n-1}\right)\cdot\vec{C}_{IJ}^{\,\Omega_r}\right|}{\left|\vec{C}_{IJ}^{\,\Omega_r}\right|}\right)^2}$$

}
Implicit_ts=false;
}
Loop over edges to:
Nodal fluxes calculation;
Δt^{Ω_r} *calculation;*
}// end-domains loop
$\Delta t = min(\Delta t^{\Omega_r})$ *;* *// explicit time-step advance*
$\Delta t_P^{n+1} = \frac{DVTOL}{\left|\Delta\vec{v}_T\right|^n}\Delta t_P^n$ *// implicit time-step advance*
$sum\_time = sum\_time + \Delta t;$
$if\,(sum\_time \geq \Delta t_P^{n+1})$
Explicit_ts=false;
else
loop over nodes:
S_w^{n+1}*calculation*
}
}
$simulation\_time = simulation\_time + \Delta t_P^{n+1}$
} // end-of-while

Figure 4.1 – MIMPES algorithm.

Equation (22) represents the variation of the velocity fields during the time range (Δt_P^n) calculated on the mean flux variation (normal fluxes) through the control volumes surfaces for all mesh control volumes. In Figure (4.1), *sum_time* accumulates all explicit time-steps within the same implicit time-step. While *sum_time* is less than Δt_P^{n+1}, both, the pressure and the velocity fields are not modified.

To avoid too large time-steps for the pressure field, which could affect the accuracy, or too small time-steps that would diminish the efficiency of the procedure even more the simulation time, the following control procedure was adopted (Hurtado et al., 2006)

$$0.75 \leq \frac{\Delta t_P^{n+1}}{\Delta t_P^n} \leq 1.25 \tag{23}$$

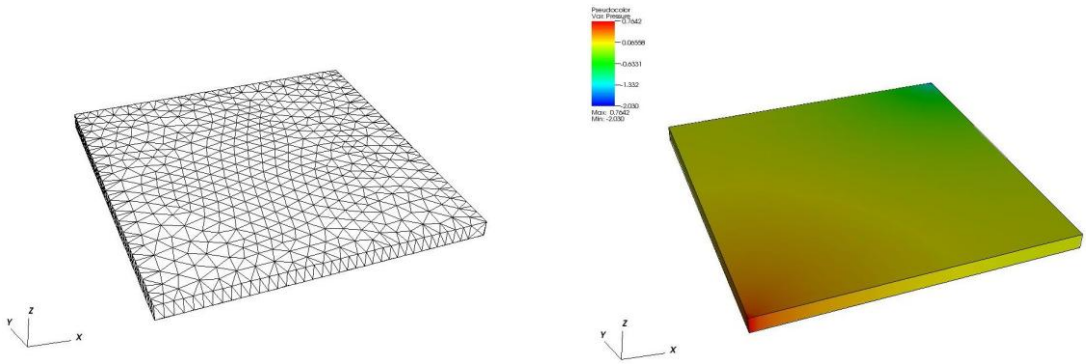
Therefore, if Δt_P^{n+1} is calculated by Eq. (21), and if it is out of the bounds determined by Eq. (23), the new implicit time-step is calculate as follows: $\Delta t_P^{n+1} = 0.75\Delta t_P^n$ or $\Delta t_P^{n+1} = 1.25\Delta t_P^n$.

In order to use this methodology in a safety way, it is necessary to check its effect on the accuracy of the numerical solutions obtained. Initially, it must be determined which values of DVTOL give the best accuracy/CPU time ratio.

5. NUMERICAL RESULTS

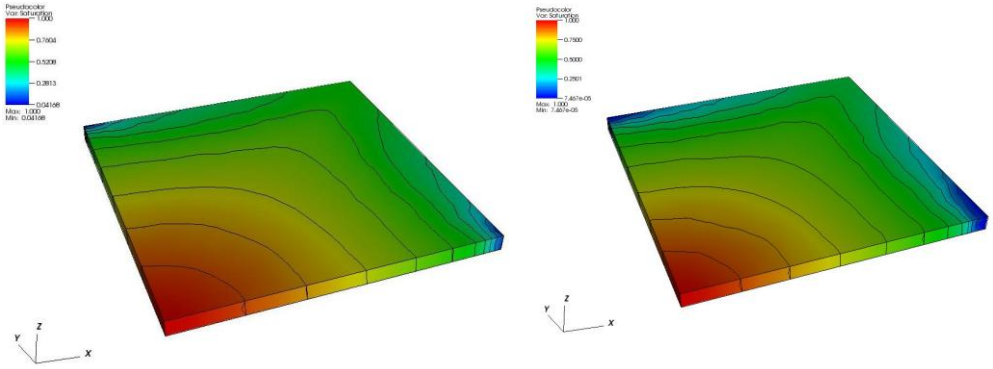
This problem, which was adapted from Durlofsky (1993) and Souza et al. (2004), consists in a ¼ of five spot problem. The porous media is assumed to be homogeneous and isotropic with $K = \underset{\sim}{I}$ throughout the whole domain. We also assume that porosity is homogeneous even though its actual value is not relevant because we are only using it to define the non-dimensional time or PVI (Pore Volumes Injected). Water and oil viscosities are, respectively, $\mu_w = 1.0$ and $\mu_o = 4.0$, therefore, the viscosity ratio (essentially the mobility ratio) is $M = (\mu_o/\mu_w) = 4.0$. Boundary conditions are no-flux at the external boundaries, $S_i = 1.0$ in the injection well, and $P_{ul} = P_{br} = 0.0$ at the upper left and bottom right corners. Durlofsky (1993) solved this problem using a combined mixed finite element (used to solve the

pressure/velocity problem) and finite volume (used to solve the saturation problem) approach. Figures (1a) and (1b) present, respectively, the computational mesh with 1236 nodes and 3892 tetrahedral elements, and the pressure field at t=1.0 PVI, obtained with the classical IMPES approach. In figures (2a) and (2b), we show the saturation fields obtained with the classical IMPES and with the MIMPES (DVTOL=10^{-1}) approaches at t=1.0 PVI. As we can clearly see, results are essentially the same. Figure (3) and (4) present, respectively, the recovered oil and the cumulative oil obtained using the classical IMPES and the MIMPES approaches with different values of DVTOL, ranging from 10^{-3} to 10^{-1}. Our results are in excellent agreement with those of Durlofsky (1993).



(a) (b)

Figure 1. ¼ homogeneous five-spot: a) Computational Mesh with 1236 nodes and 3892 tetrahedral elements; b) Pressure Field at t=1.0 PVI obtained with the classical IMPES method.



(a) (b)

Figure 2. ¼ homogeneous five-spot problem: a) Saturation field obtained with the classical IMPES method at t=1.0 PVI; b) Saturation field obtained with the MIMPES method at t=1.0 PVI, with DVTOL=10^{-1}.

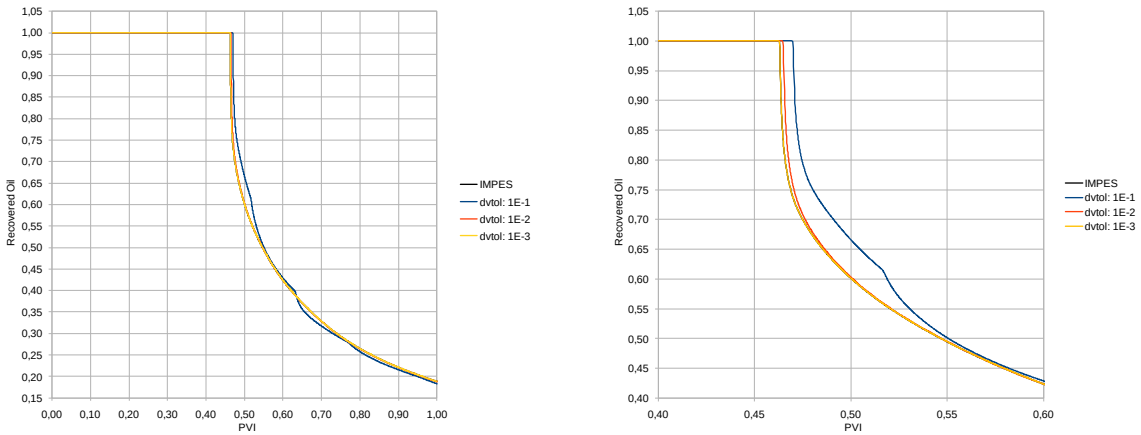


Figure 3. Recovered oil for the ¼ homogeneous five-spot problem with the classical IMPES and with the MIMPES for DVTOL raging from 10^{-3} to 10^{-1} (zoom on the right of the figure).

Figure (5) compares the relative CPU time and the relative difference of the cumulative oil for the classical IMPES and the MIMPES approaches for different values of DVTOL. Considering the results obtained with the classical IMPES method as our reference, it can be clearly seen that, small values of DVTOL produce more accurate results, while higher DVTOL values produce the poorest results. This is expected, since smaller values of DVTOL make the MIMPES scheme gets closer to the classical IMPES approach (i.e. with more pressure/velocity updates).

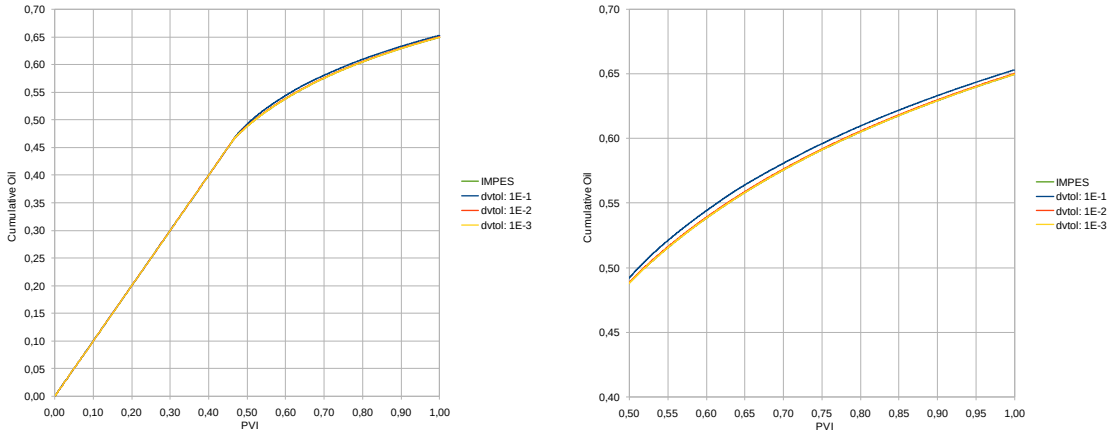


Figure 4. Cumulative oil for the ¼ homogeneous five-spot problem with the classical IMPES and with the MIMPES for DVTOL raging from 10^{-3} to 10^{-1} (zoom on the right of the figure).

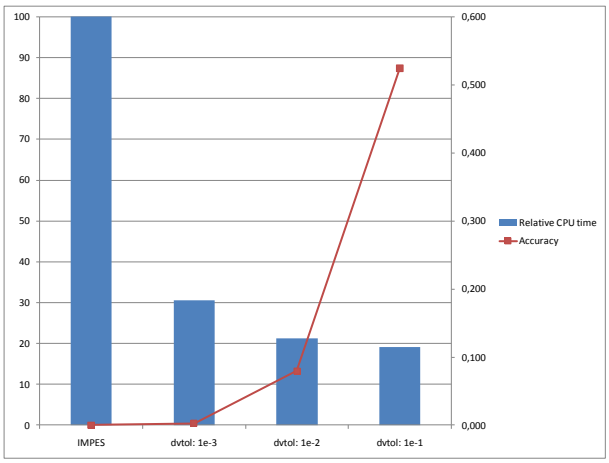


Figure 5. CPU time and relative difference of the cumulative oil between the classical IMPES and MIMPES for DVTOL ranging from 10^{-3} to 10^{-1}.

6. CONCLUSIONS

In the present paper, we have briefly presented a node-centered, edge-based, higher order finite volume method with a "Modified Implicit Pressure, Explicit Saturation" (MIMPES) which is capable to model the 3-D incompressible and immiscible two-phase flow of water and oil in heterogeneous and anisotropic porous media. The solution of the pressure equation is performed by using a variation of the two step Crumpton's method (Crumpton, 1997) and the saturation equation is solved by a MUSCL-type method proposed by Woodfield et al. (2005), with a multidimensional limiter that resembles the one proposed by Zalezak (1979). We have also used a MIMPES approach (Hurtado et al. 2006) that allows for larger time steps for the pressure/velocity problem while the explicit solution of the saturation equation, which is still restricted by the CFL condition, is performed many times before the pressure/velocity fields is updated. This approach produces very acceptable results with a considerable reduction in CPU time. Through a very simple benchmark problem, we have showed that the combination of these different techniques allow for a very accurate simulation at a reasonably computational cost. In the near future, we intend to implement a fully implicit higher order, edge-based finite volume procedure in order to solve more complex and demanding problems.

7. ACKNOWLEDGEMENTS

The authors would like to thank the following agencies: "Coordenação de Aperfeiçoamento de Pessoal de Nível Superior (CAPES)", "Agência Nacional de Petróleo (ANP PRH-26)" and the "Conselho Nacional de Desenvolvimento Científico e Tecnológico (CNPq)", for their financial support during the development of this work.

8. REFERENCES

Carvalho, D.K.E., Lyra, P.R.M., Willmersdorf, R.B., Araújo, F.D.S., 2005, "An Unstructured Edge-Based Finite Volume Formulation for Solving Immiscible Two-Phase Flows in Porous Media" Communications in Numerical Methods in Engineering, Vol. 21, pp. 747-756.

Carvalho, D.K.E. ; Willmersdorf, R.B. ; Lyra, P.R.M., 2009, "Some results on the accuracy of an edge-based finite volume formulation for the solution of elliptic problems in non-homogeneous and non-isotropic media", International Journal for Numerical Methods in Fluids, DOI: 10.1002/fld.1948.

Chen, Z., Huan, G., Li, B., Li, W., Espin, D., 2002, "Comparison of Practical Approaches to Reservoir Simulation", 2nd Meeting on Reservoir Simulation, Buenos Aires, Argentina (in CD-ROM).

Crumpton, P.I., Moinier P., Giles, M.B.T.J., 1997, "An Unstructured Algorithm for High Reynolds Number Flows on Highly Stretched Grids", In: Numerical Methods in Laminar and Turbulent Flow, Taylor C, Cross J.T. (Eds). Pineridge Press: Swansea, pp. 561–572.

Durlofsky, L.J.A., 1993, "Triangle Based Mixed Finite Element-Finite Volume Technique for Modeling Two Phase Flow Through Porous Media", Journal of Computational Physics, v. 105, p. 252-266, 1993.

Ewing, R.E., 1983, "The Mathematics of Reservoir Simulation", Siam, Philadelphia.

Helmig R, Huber R., 1998, "Comparison of Galerkin-Type Discretization Techniques for Two-Phase Flow Problems in Heterogeneous Porous Media", Advances in Water Resources, Vol. 21, pp. 697-711.

Hurtado, F. S. V., Cordazzo, J., Maliska, C. R., Silva, A. F. C., 2006, "Advanced Numerical Techniques for Improving Reservoir Simulation", Rio Oil and Gas, 2006, in CD ROM.

LeVeque, R. J., 1992, In: Numerical Methods for Conservation Laws. Berlin, Birkhauser.

Luo, H., Baum, J.D., Löhner, R., 1995, "An Improved Finite Volume Scheme for Compressible Flows on Unstructured Grids", Technical Report, AIAA Paper, pp. 95-0348.

Lyra, P.R.M., Morgan, K., 2002, "A Review and Comparative Study of Upwind Biased Schemes for Compressible Flow Computations: Part III: Multidimensional Extension on Unstructured Grids", Archives of Computational Methods in Engineering, Barcelona - Espanha, Vol. 9, pp. 207-256.

Müller D, Giles MB. Edge-based multigrid schemes for hybrid grids. 6th ICFD Conference on Numerical Methods for Fluid Dynamics, Oxford, UK, 1998; 1-7.

Rees I. Development of an edge-based finite volume solver for porous media applications. Department of Civil Engineering – University of Wales/Swansea- UK, Ph.D. thesis, 2004.

Seol, E. S. FMDB: Flexible Distributed Mesh Database For Parallel Automated Adaptive Analysis, Doctor Of Philosophy, Faculty of Rensselaer Polytechnic Institute, 2005.

Silva, R. S., 2008, "Simulação de Escoamento Bifásico Óleo-Água em Reservatórios de Petróleo usando Computadores Paralelos de Memória Distribuída", Tese de Doutorado, DECIV-UFPE.

Smith MI, Griffiths DV. Programming the finite element method. John Wiley & Sons, Chichester, 2001.

Souza, D. A. F.; Coutinho, A.L.G.A.; Alves, J.L.D., 2004 "Three Dimensional Finite Element Solutions of Immiscible Two-Phase Flow Through Porous Media", Congresso Íbero-Latino Americano de Métodos Computacionais em Engenharia (CILAMCE 2004), (Em CD-ROM).

Verma S, Aziz K., 1996, "Two and Three Dimensional Flexible Grids For Reservoir Simulation.", 5th European Conference on the Mathematics of Oil Recovery (ECMOR V), 1-14, Leoben, Austria.

Woodfield, P.L., Suzuki, K., Nakabe, K., 2004, "A Simple Strategy for Constructing Bounded Convection Schemes for Unstructured Grids", International Journal for Numerical Methods in Fluids, Vol. 46, pp. 1007-1024.

Zalesak, S.T., 1979, "Fully multidimensional flux-corrected transport algorithms for fluids." J. Comput. Phys. v31. 335-362.

5. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.