NUMERICAL SIMULATION OF OIL-WATER DISPLACEMENTS IN POROUS MEDIA USING A MESH ADAPTIVE FINITE VOLUME FORMULATION

Carvalho, D. K. E., dkarlo@uol.com.br
Núcleo de Tecnologia - Centro Acadêmico do Agreste (CAA)
Universidade Federal de Pernambuco (UFPE)
Rodovia BR 104, KM 62, S/N, Pólo Comercial Nova Caruaru,
Caruaru-PE, Brasil, CEP: 55002-970

Luna, B. G. B.[1], Willmersdorf, R. B.[2], Lyra, P. R. M.[3]
[1]bruno_bl@yahoo.com.br,[2]ramiro@willmersdorf.net,[3]prmlyra@ufpe.br
Departamento de Engenharia Mecânica (DEMEC)
Centro de Tecnologia e Geociências (CTG)
Universidade Federal de Pernambuco (UFPE)
Av. Acadêmico Hélio Ramos S/N Cidade Universitária, Recife – PE, Brasil, CEP: 50.740-470

Queiroz, J. P. S, joaosqueiroz@hotmail.com
Departamento de Engenharia Química (DEQ)
Centro de Tecnologia e Geociências (CTG)
Universidade Federal de Pernambuco (UFPE)
Av. Acadêmico Hélio Ramos S/N Cidade Universitária, Recife – PE, Brasil, CEP: 50.740-470

Abstract. *Unstructured mesh based discretization techniques can offer advantages relative to standard finite difference approaches which are still largely used to model flux flow through porous media (e.g oil-water displacements), due to their flexibility to model complex geological features and due to their capacity to incorporate mesh adaptation techniques. In this paper, we consider an unstructured edge-based finite volume formulation (FV) which is used to solve the elliptic pressure equation and the non-linear hyperbolic equation that arises in oil-water displacements problems when IMPES (IMplicit Pressure Explicit Saturation) techniques are used. The IMPES method is a segregated type method in which the flow equations are manipulated in order to produce an elliptic pressure equation which is solved implicitly and a hyperbolic type saturation equation, which is solved explicitly. The numerical formulation includes the use of an edge-based high order upwind type method to deal with the "non-viscous" terms of the Bucklet-Leverett (saturation) equation. In order to improve the overall accuracy of the solution at a reasonable computational cost, we have used a h-type mesh adaptive procedure. An "a-posteriori" error estimator based on gradient recovery is used to control the adaptive process. A simple two dimensional model example is solved in order to show the potentiality of the presented formulation.*

Keywords*: unstructured meshes, edge-based data structure, h-type mesh adaptation, finite volume method*

1. INTRODUCTION

In the present paper, we consider an edge-based unstructured finite volume formulation using node centered median dual control volumes. This FV formulation is used to solve the partial differential equations resulting from the modeling of the immiscible two-phase fluid flow of oil and water when the IMPES (IMplicit Pressure Explicit Saturation), approach is used (Ewing, 1983; Carvalho *et al.*, 2005; Hurtado *et al.*, 2005; Carvalho *et al.*, 2006; Hurtado *et al.*, 2006). 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. The edge-based data structure allows for the implementation of different types of upwind biased discretizations in the context of 2-D and 3-D unstructured meshes (Lyra and Morgan, 2002; Woodfield *et al*., 2004; Carvalho *et al*., 2005). Here, in order to account for the non-viscous term appearing in the saturation equation, we use a high order upwind type method adapted for unstructured meshes. The edge-based finite volume formulation is very flexible and efficient, being similar to the edge-based finite element formulation when linear triangular (tetrahedral in 3-D) elements are employed.

The use of automatic mesh adaptation is very attractive because it allows for a better representation of the saturation fronts occurring in immiscible flows and because it can yield a more accurate representation of the fluid flow near wells, at a reasonable computational cost. To perform the mesh adaptation, we have used a local mesh embedding or h-type adaptation, in which the elements of the triangular mesh are either subdivided or grouped according to an "a-posteriori" error estimator based on a global gradient recovery technique.

2. MATHEMATICAL FORMULATION

The governing equations for incompressible, immiscible two-phase displacement of oil by water through rigid porous media are obtained by combining Darcy's Law with the mass conservation equation for each phase (Ewing 1983; Chen *et al*., 2002; Carvalho *et al*., 2005; Carvalho *et al*., 2006).

Assuming that the phase velocities obey the Darcy's law, and by ignoring gravitational effects, we can write for phase *i*,

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

where *i = o (oil)* and *w (water)*, $\underset{\sim}{K}$ denotes the absolute permeability tensor of the rock, and the phase mobility is defined as,

$$\lambda_i = \frac{k_i}{\mu_i} \tag{2}$$

with k_i being the phase relative permeability and μ_i the phase viscosity. We will also assume incompressible fluids and porous media. We will also neglect 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. The 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 volume occupied by this phase. Due to this last definition, we can write the constitutive relantionship,

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

Combining Eq.(1) to Eq. (3) and after some algebraic manipulation we obtain the following pressure equation,

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

where, $\lambda = \lambda_o + \lambda_w$ is the total fluid mobility, $\underset{\sim}{K}$ is the permeability tensor, $\vec{v} = \vec{v}_o + \vec{v}_w = -\lambda \underset{\sim}{K} \nabla P$ 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.

In 2-D, the porous media permeability $\underset{\sim}{K}$ which is, in general, a full tensor, can be written in Cartesian coordinates as,

$$\underset{\sim}{K} = \begin{pmatrix} K_{xx} & K_{xy} \\ K_{yx} & K_{yy} \end{pmatrix} \tag{6}$$

where, for the pressure equation to be elliptic, we further assume that $K_{xx} K_{yy} \geq K_{xy}^2$.

By introducing the fractional flow function $f_i = \lambda_i / \left(\lambda_o + \lambda_w\right)$, we can also derive a non-linear hyperbolic equation for the water saturation, which can be written as,

$$\phi \frac{\partial S_w}{\partial t} + \nabla \cdot \vec{F}_w = Q_w \tag{7}$$

In Equation (7), the term $\vec{F}_w = f_w \vec{v}$ is the flux function which is a non-linear function of the water phase saturation, i.e., $\vec{F}_w = \vec{F}_w \left(S_w\right)$. It is worthy noting that the pressure and saturation fields are connected through the total velocity $\vec{v}$.

Therefore, the accuracy of the velocity field, which is directly obtained from the pressure field, highly influences the solution of the hyperbolic saturation equation.

3. NUMERICAL FORMULATION

As mentioned previously, in the present work we have adopted a node centered median dual control volume 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; Crumpton *et al*., 1997; Sorensen, 2001; Lyra *et al*., 2004, Rees *et al*., 2004; Carvalho *et al*., 2005). 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.

Even though, there is 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 orthogonal quadrilateral (2D) and hexahedral (3D) meshes. Therefore, extra care must be taken when using different element types, especially when considering 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. 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.

3.1. Implicit pressure equation

In order to obtain our discrete equations, we can write,

$$\nabla \cdot \vec{v} = Q \tag{8}$$

Integrating (8) over the domain and using the divergence theorem, yields,

$$\int_{\Gamma} \vec{v} \cdot \vec{n} \, \partial\Gamma = \int_{\Omega} Q \, \partial\Omega \tag{9}$$

After some algebraic manipulation, for a node *I* of the mesh, we can write the discrete form of Eq. (9) as,

$$\sum_{L_{I(\Omega)}} \vec{v}_{IJ_L}^{\,\Omega} \cdot \vec{C}_{IJ_L} + \sum_{L_{I(\Gamma)}} \vec{v}_{IJ_L}^{\,\Gamma} \cdot \vec{D}_{IJ_L} = Q_I V_I \tag{10}$$

In Equation (10), summations are performed over the edges $\left(L_I\right)$ connected to node *I*. The upper index Ω represents approximations on the middle of edges IJ_L connected to node *I,* and Γ refers to boundary edges only. $\vec{v}_{IJ_L}$ stands for the mid-edge velocity and V_I is the volume of the CV surrounding node *I*. The geometrical coefficients $\vec{C}_{IJ_L}$ and $\vec{D}_{IJ_L}$ are defined as,

$$\begin{aligned} \vec{C}_{IJ_L} &= A_{K+1}\vec{n}_{K+1} + A_K\vec{n}_K \\ \vec{D}_{IJ_L} &= A_L\vec{n}_L \end{aligned} \tag{11}$$

In Equation (11), $A_K = TL_K$, $A_{K+1} = TL_{K+1}$ and $A_L = TL_L$ are the areas of the control volume face associated to the normals $\vec{n}_K$, $\vec{n}_{K+1}$ and $\vec{n}_L$, respectively, and T is the thickness of the domain.

In order to approximate the mid-edge gradients/velocities required in Eq. (10), different strategies can be devised (Svärd and Nordström, 2003). A classical approach involves using a simple two point approximation in which mid-edge velocities are formally second order accurate only if the media is isotropic and the straight lines that connects two adjacent nodes and the control volume faces are orthogonal to each other as in the case of the Voronoi tessellations (Edwards and Rogers, 1998). Schemes using such approaches are equivalent to the so called control volume finite difference methods (CVFD). In the present work, we have used a different approach which was originally devised by Crumpton *et al*. (1997) for the discretization of diffusion terms of the Navier-Stokes equations. In order to obtain the final discrete system of equations, we first determine nodal gradients as functions of the discrete pressure field and then,

we use these gradients to compute the elliptic terms in a second step. In this approach, mid-edge gradients/velocities are split in two parts, the gradient component parallel to the edge is obtained via a two point finite difference approach, while the gradient component normal to the edge is obtained via the projection of the arithmetic mean of the gradients of the nodes belonging to the edge, which were computed in the first step. Further details about the method and the treatment of non-homogeneous and anisotropic materials can be found in Crumpton *et al*. (1997), Sorensen (2001), Lyra *et al*. (2004), Rees *et al*. (2004), Carvalho *et al*. (2005).

3.2. Explicit saturation equation

In petroleum reservoir simulators, the common way to discretize the advective term that characterize the hyperbolic saturation equation is to use classical first order upwind type methods which are 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 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 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 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 over and undershoots in the numerical solution. Below, we present a brief description of the scheme. For further details, see Woodfield *et al*. (2005).

By integrating Eq. (7) 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 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,

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

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. During the adaptive process, close to elements with high aspect ratio, 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.

4. MESH ADAPTATION STRATEGY

The accuracy of a numerical simulation depends on the order of the approximation and on the distribution of the nodes along the computational domain. The density, the shape and the gradation of the elements depend, among other things, upon the numerical method used and upon the geometric characteristics of the problem. Usually, based on some kind of knowledge of the solution behavior or on some error estimate a mesh adaptation procedure can be devised.

The mesh adaptation procedure used in the present work requires several steps. The computational codes utilized are fully independent and we have chosen to build a macro-code which controls all the adaptation procedure, transferring the necessary information among the different programs (Lyra *et al.*, 2002). In Figure 1 we show a flux diagram which shortly describes the steps of the adaptation algorithm. For transient problems, after each time step, or a pre-defined number of time steps, the error analysis is performed and when necessary the time integration is interrupted. During the transient analysis, when the error tolerance is achieved a new mesh is built and the solution of the previous time level t^n is interpolated to the new mesh generated and then, the time integration proceeds. We discard the solution at time level t^{n+1} in order to avoid the use of a solution which does not satisfy the established error criteria avoiding the error accumulation and the deterioration of the solution (Araújo *et al*., 2004).

4.1. Mesh generation

In order to perform the spatial discretization we have used a computational program capable to generate triangular, quadrilateral and mixed two-dimensional meshes (Lyra and Carvalho, 2006), even though, in the present paper, we have only used triangular meshes. This program can generate iso and anisotropic (i.e. directional) meshes over multiple domains. The triangulation of the domain is performed through an "Advancing Front Technique", in which the control mesh parameters (i.e. nodal spacing, direction and stretching factor) are interpolated from the values introduced through a background mesh and the quadrilateral and mixed meshes are obtained through an indirect approach which involves the merging and splitting of triangular elements. This approach is completely described in Lyra and Carvalho, (2006) and references thein.

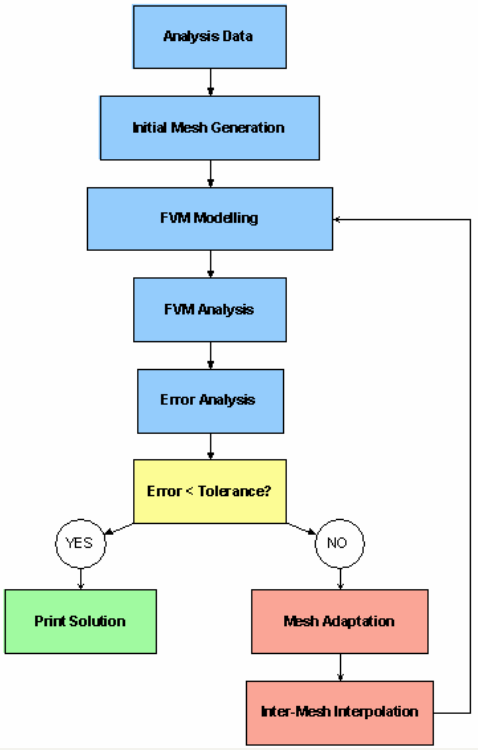


Figure 1. Flux diagram describing the steps of the adaptation algorithm.

4.2. Finite volume model

After the spatial discretization, the next step consists in the generation of the finite volume model, which also includes the proper definition of initial and boundary conditions. The mesh topological data (nodes, edges and elements) are associated to the geometrical entities (points, curves and domains). This allows for the association of the initial and boundary conditions to the geometrical entities. This is extremely important due to the fact that, in an adaptive procedure, each new created mesh is associated to a new numerical model.

4.3. Error analysis

The error analysis tool controls all the adaptive procedure. It indicates the values of the parameters used to build the new mesh and it also defines the stop criteria of the adaptive process. A posteriori error estimator based on the gradient projection is used. The main idea of this error estimator is to compute the difference between the approximate discontinuous gradient field and the approximate continuous gradient field. The latter is computed through the "Global Variational Recovery" technique (Zienkiewicz and Zhu, 1987).

In our adaptation strategy we compute the mesh parameters through an error analysis in which two error estimates are used. In the first one, the global solution error is computed taking into account all the elements of the mesh and, in the second one, we exclude the elements which can not be refined based in the adaptation criteria as it would create elements with spacing smaller than the allowed one (Araújo *et al*. 2004).

In two-phase fluid flows, multiple choices can be made in the definition of the variable that dictates the mesh adaptation procedure, such as the saturation, the pressure or the velocity field. In the present work, we have chosen the somehow conservative choice of using both, saturation and pressure as the control variables. Each time the error analysis is performed, the global error is computed for both fields and the mesh is adapted whenever the error for one of the two variables is over the prescribed tolerance. This choice is certainly conservative and further studies are necessary

to improve it, but it certainly guarantees that the mesh will be improved near wells and along the saturation shocks where both variables present their highest gradients, which are precisely the spots where a better resolution of two fields is of utmost importance.

4.4. H-type mesh adaptation

In this mesh adaptation strategy, the meshes are automatically refined in regions of coarse resolution presenting discretization errors above the tolerance and they are derefined in regions where the errors are below a prescribed tolerance. Bellow, we shortly describe the procedure of the h-type adaptation,

1. Determine the refinement level for each element;
2. Bound the interelement difference of level in an unity;
3. Assure good mesh regularity;
4. Handle irregular nodes through transition elements;
5. Create a new mesh.

The degree of the element refinement/derefinement, i.e. how much an element must be subdivided or grouped is determined through an error analysis and it is up to the user to limit or not the subdivision or grouping of elements in a single stage. Using the error indicator for each element and the global error estimate together with the information of the theoretical behavior of the convergence rate of the solution and the equal distribution of errors (Zienkiewicz and Zhu, 1987), is possible to estimate the distribution of the element sizes necessary to achieve the prescribed tolerance. These estimated sizes are compared to the current mesh, defining the refinement degree, which is set positive for element subdivision and negative for element grouping. Our "standard" pattern of subdivision splits triangles into four triangular elements through the creation of intermediate nodes on the middle of the edges, for further details, see Araújo *et al*. (2004) and references therein.

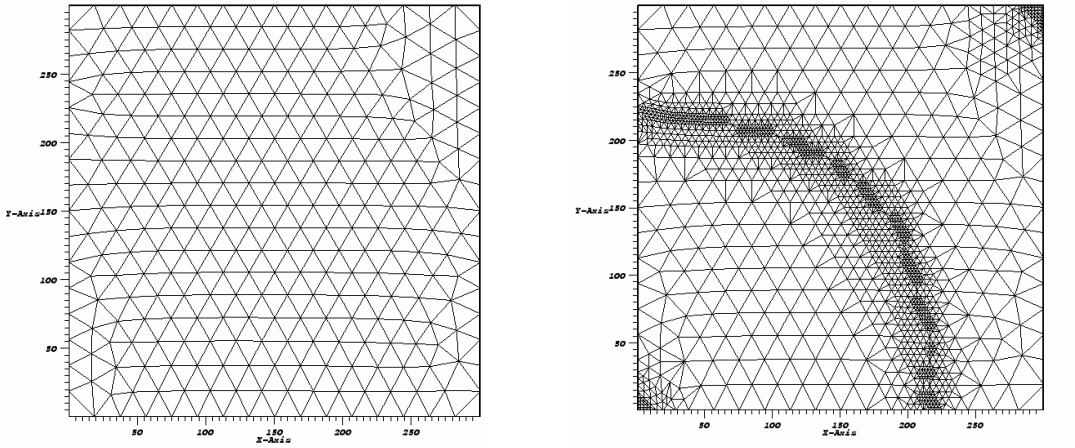
In order to simplify the process of creation of a final consistent mesh, we used two criteria to adjust the degree of each element refinement to build the new entities of the mesh. The first criterion adjusts the element levels in a way that the biggest difference among the degrees of refinement of the neighbor elements is a unity. The second criterion aims at the formation of a more regular mesh. Therefore if an element has more than two neighbors with a degree bigger than its own level we add one unity to its degree.

At the end of the three first steps of the general algorithm we have a mesh with irregular nodes which are eliminated through transition elements of the same kind, subdividing an element in other two. The last step, involves the building of the connectivity matrices, the boundary edges and the coordinates which describe the new mesh.

5. EXAMPLE

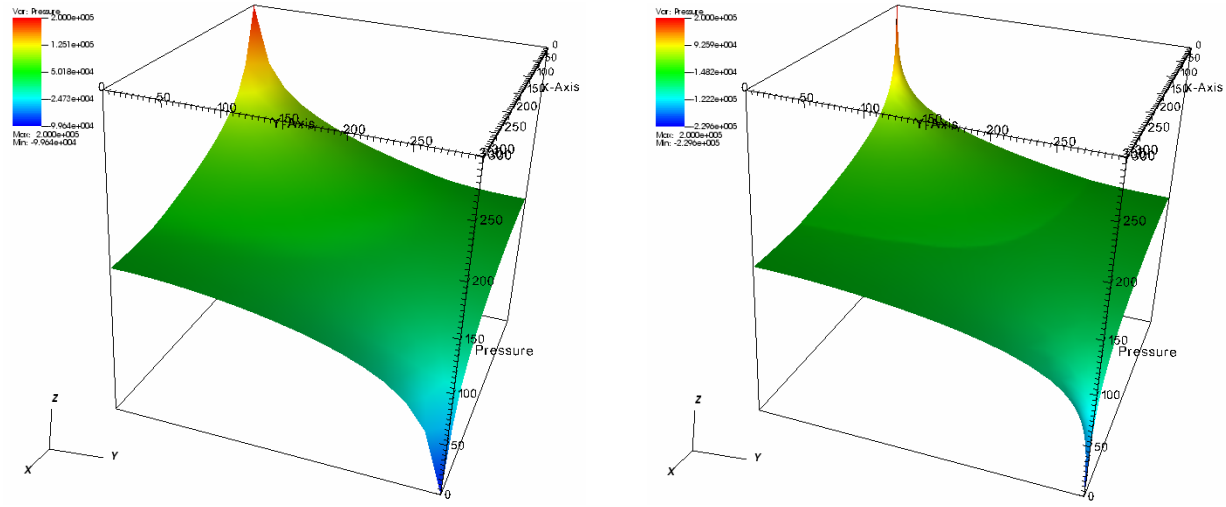
5.1. Homogeneous ¼ of five-spot water flood problem

This problem, which was adapted from Helmig (1997), consists in a ¼ of five spot problem. Residual saturations are $S_{rw} = S_{ro} = 0.0$ and fluid properties are $\rho_w = \rho_o = 1000\ kg/m^3$ and $\mu_w = \mu_o = 0.001\ kg/(m.s)$, the absolute permeability is equal to $\underset{\sim}{K}_2 = 10^{-7}\,\underset{\sim}{I}\,\mathrm{m}^2$ and porosity is constant with $\phi = 0.2$ throughout the whole domain. Pressure and saturation boundary conditions are, $p = 2.10^5\,Pa$ and $S_w = 1$ at the injection well and $Q = -10.368\,m^3/d$ at the producer. We have used a quadratic relative permeability-saturation relationship (Helmig, 1997). For this case, we have used a tolerance of 0.45 for the global error and 0.43 for the global error computed after eliminating the elements that have achieved the minimum spacing allowed. In Figures 2a and 2b we show, respectively, a coarse mesh with 310 nodes and 554 triangular elements, which was used for both, the complete solution of the problem, and as the initial mesh of the adaptive procedure, and the adapted mesh with 1538 nodes and 2966 triangular elements, at t = 650 days. In Figures 3a and 3b, we present, respectively, the "extruded" pressure field at t = 650 days using the coarse mesh and the adapted mesh. As it can be observed, it is clear that the adapted mesh provides a much better resolution of the pressure field, and consequently, of the velocity field, near to the injection and production wells.



(a) (b)

Figure 2. a) Initial coarse mesh; b) Adapted mesh at t = 650 days.

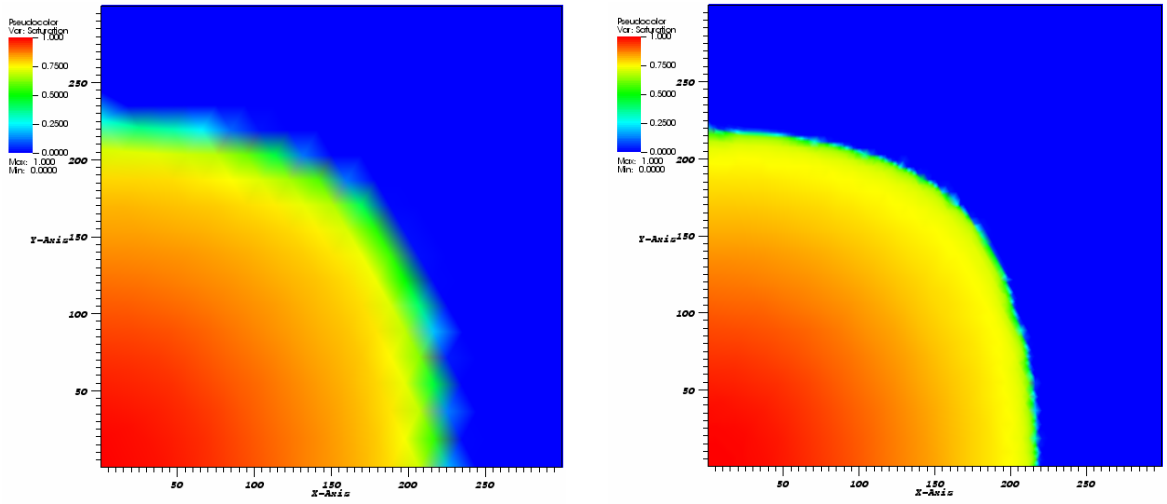


(a) (b)

Figure 3. Extruded pressure field at t = 650 days: a) using the coarse mesh; b) using the adapted mesh.

In Figures 4a and 4b, we present, respectively, the saturation fields obtained with the coarse mesh and with the adapted mesh at t = 650 days. As it can be seen, using the adapted mesh the saturation front is much better defined (i.e. less spread) than the one obtained with the coarse mesh.

Figure 5 presents the behavior of the relative error during the first 160 steps of the analysis. It must be emphasized that the graphic stands for the maximum relative error obtained taking into account the pressure and the saturation fields at each adaptation step. It is quite interesting to observe that the adaptive procedure keeps the global error quite close to the target value (i.e. tolerance) of 0.45 throughout the analysis. It is worthy mentioning that, to obtain a similar result in terms of the relative error with a mesh of constant spacing, we had to use a mesh with 13874 nodes and 27310 triangular elements, which is a mesh with more than 9 times the number of nodes and elements of the equivalent adapted mesh.



(a) (b)

Figure 3. Saturation field at t = 650 days: a) using the coarse mesh; b) using the adapted mesh.

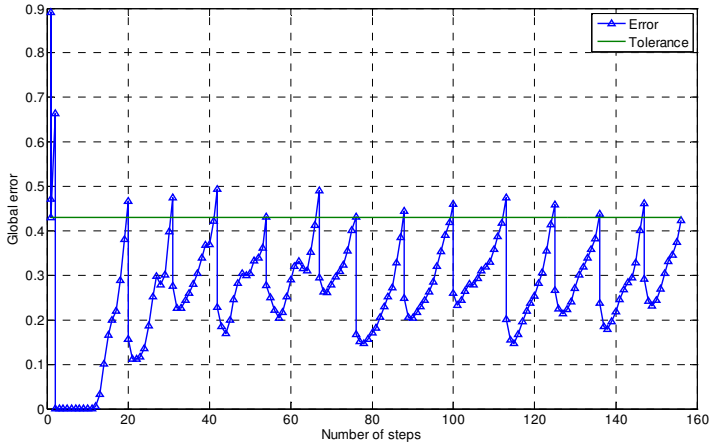


Figure 4. Global error behavior versus time steps.

6. CONCLUSIONS

In the present paper we have briefly described an h-type mesh adaptive procedure coupled with an edge-based node centered fully finite volume formulation (EBFV), which were used to simulate the two-phase flow of oil and water in porous media. The elliptic pressure equation was solved using a modification of Crumpton's two step approach that is capable of handling heterogeneous and non-isotropic materials (not presented in the present paper) and the hyperbolic saturation equation was solved through a high order upwind type approach capable to deal with multidimensional problems. The adaptive procedure was guided by an "a-posteriori" error indicator based in a "Global Variational Recovery" technique of Zienkiewicz and Zhu. Apart from the solver and the adaptive tools (i.e. h-type adaptation, error analysis and interpolation routines), the computational package includes a two-dimensional mesh generator which is capable to generate triangular, quadrilateral or mixed meshes, even though only triangular meshes were used. Despite of the intensive read and write disk operations associated to the information exchange among different parts of the computational package, the finite volume procedure together with the mesh adaptive tools are capable of producing very accurate results at a reasonably computational cost, being quite useful for the academic purpose of studying different solvers, interpolation techniques, mesh adaptation strategies, etc. In the near future, we intend to investigate the use of different variables to control the mesh adaptive procedure such as the total velocity or the combination of pressure and saturation in a single error norm. Besides, we intend to investigate the use of edge-based error indicators in order to further improve the computational efficiency. A simple model problem was solved in order to show the potentiality of the proposed adaptive finite volume formulation.

7. ACKNOWLEDGEMENTS

The authors would like thank the National Research Council (CNPQ) and the National Petroleum Agency (ANP) for the financial support.

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., 2006, "A Node-Centered Finite Volume Formulation For The Solution of Two-Phase Flows In Non-Homogeneous Porous Media" *International Journal for Numerical Methods in Fluids*, Vol. 53, pp. 1197-1219.

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.

Edwards, M.G., Rogers, C.F., 1998, "Finite Volume Discretization with Imposed Flux Continuity for the General Tensor Pressure Equation", *Computational Geosciences 2*, Vol. 2, pp. 259-290.

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

Helmig, R., 1997. Multiphase Flow and Transport Processes in the Subsurface. Springer-Verlag.

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., Maliska, C. R., Silva, A. F. C., Cordazzo, J., 2005, "Advanced Numerical Techniques for Improving Reservoir Simulation", *XXVI Iberian Latin-American Congress on Computational Methods in Engineering* (CILAMCE 2005), in CD ROM.

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.

Hirsch, C., 1990, In: *Numerical Computation of Internal and External Flows: Volume II, Computational Methods for Inviscid and Viscous Flows*. New York, Wiley & Sons..

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., Carvalho, D.K.E. de, Willmersdorf, R.B., Almeida, R.C., 2002, "Transient Adaptive Finite Element Analysis of Compressible Flows", In*: Fifth World Congress on Computational Mechanics (V WCCM)*, Viena, Vol. 1 pp. 1-11.

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.

Lyra, P.R.M., Lima, R.C.F. De, Guimarães, C.S.C., Carvalho, D.K.E. de, 2004, "An Edge-Based Unstructured Finite Volume Procedure for the Numerical Analysis of Heat Conduction Applications", *Journal of the Brazilian Society of Mechanical Engineering*. Vol. 26, pp. 160-169.

Lyra, P.R.M., Carvalho, D.K.E. de, 2006, "A Computational Methodology for Automatic Two-Dimensional Anisotropic Mesh Generation and Adaptation", *Journal of the Brazilian Society of Mechanical Science and Engineering*, Vol. 28, pp.399-412.

Rees, I., Masters, I., Malan, A.G., Lewis, R.W., 2004, "An Edge-Based Finite Volume Scheme for Saturated-Unsaturated Groundwater Flow", *Computer Methods in Applied Mechanics*, Vol. 193, pp. 4741-4759.

Sorensen, K.A., 2001, "A Multigrid Accelerated Procedure for the Solution of Compressible Fluid Flows on Unstructured Meshes", Department of Civil Engineering – University Of Wales/Swansea- Uk, Ph.D. Thesis C/Ph/161/01.

Araújo, F. S. de, Lyra, P.R.M., Carvalho, D.K.E. de, 2004, "Um Estudo Comparativo de Dois Procedimentos de Adaptação de Malhas ao Resolver Problemas Modelos de Reservatórios Através de uma Formulação do MVF com Estrutura de Dados por Aresta", Congresso Nacional de Engenharia Mecânica (CONEM 2004), in CD-ROM.

Svärd, M, Nordström, J., 2003, "A Stable and Accurate Summation by Parts Finite Volume Formulation of the Laplacian Operator". *Uppsala Universitet, Institutionen för Informationsteknologi, Technical Report 003*, pp. 1-36.

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.

Zienkiewicz, O.C., Zhu, J.Z., 1987, "A Simple Error Estimator and Adaptive Procedure for Pratical Engineering Analisys", *International Journal for Numerical Methods in Engineering*, 24:337-357.

9. RESPONSIBILITY NOTICE

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