A FIRST STEP TOWARDS A PETROLEUM RESERVOIR SIMULATOR USING AN EDGE-BASED UNSTRUCTURED FINITE VOLUME FORMULATION

Darlan K. E. De Carvalho[1], Paulo R. M. Lyra[2], Ramiro B. Willmersdorf[2].

[1] Universidade Federal de Pernambuco, Centro de Tecnologia e Geociências, Departamento
de Engenharia Civil, Av. Acadêmico Hélio Ramos s/n, Cidade Universitária,
darlan@demec.ufpe.br.
[2] Universidade Federal de Pernambuco, Centro de Tecnologia e Geociências, Departamento
de Engenharia Mecânica, Av. Acadêmico Hélio Ramos s/n, Cidade Universitária,
prmlyra@demec.ufpe.br, rbw@demec.ufpe.br.

Resumo – No presente trabalho nós apresentamos alguns aspectos dos problemas matemáticos associados ao escoamento multifásico em reservatórios de petróleo. Além disso, nós discutimos de forma breve, as ferramentas computacionais que estamos desenvolvendo com o objetivo de resolver adequadamente as equações diferenciais parciais (EDPs) características dos problemas de simulação de reservatórios. Tecemos alguns comentários a respeito do método dos volumes finitos em malhas não-estruturadas com estrutura de dados por aresta. Alguns exemplos modelo bidimensionais são resolvidos com o objetivo de mostrar o potencial da formulação utilizada. Adicionalmente, fazemos alguns comentários a respeito de adaptação de malhas, do método "multigrid" e sobre a utilização de computadores paralelos (clusters de computadores pessoais), os quais são ferramentas que pretendemos utilizar num futuro próximo.

Palavras-Chave: Volumes Finitos; Malhas Não-Estruturadas; Estrutura de Dados por Aresta, Simulação Numérica de Reservatórios.

Abstract – In the present paper, we present some aspects of the mathematical problem of multiphase flow through petroleum reservoirs. Besides, the numerical tools we are developing in order to solve properly the non-linear partial differential equations (PDEs) that arise in numerical reservoir simulation are briefly discussed. We also comment on the utilization of an edge based unstructured finite volume formulation. Some bidimensional model examples are solved in order to show the potential of the formulation utilized. Additionally, we make some remarks about mesh adaptation, multigrid methods and the utilization of parallel computers (clusters of personal computers), which are tools we intend to use in the near future.

Keywords: Finite Volume, Unstructured Meshes, Edge-Based Data Structure, Numerical Reservoir Simulation.

1. Introduction

Naturally occurring hydrocarbon systems found in petroleum reservoirs are mixtures of organic compounds, which exhibit multiphase behavior over wide ranges of pressures and temperatures. These hydrocarbon accumulations may occur in the gaseous, liquid and solid states or in various combinations of gas, liquid and solid phases. These differences in phase behavior, coupled with the physical properties of reservoir rock that determine the relative ease with which gas and liquid are transmitted or retained, result in many diverse types of hydrocarbon reservoirs with complex behaviors. Frequently, petroleum engineers have the task to study the behavior and characteristics of a petroleum reservoir and to determine the course of future development and production that would maximize profits.

Nowadays, in order to complement and improve the accuracy of the older prediction methods (e.g. experimental, analogs, etc.), numerical methods are being widely used as tools to predict the behavior of multiphase flow through reservoirs

Due to several factors (robustness, easiness of programming, etc), the finite difference method (FDM) is usually utilized in reservoir analysis. On the other hand much effort has been recently put in methods that allows a better treatment of the complex geometries that characterize petroleum reservoirs. In this context, the adoption of methods able to deal with unstructured meshes is very attractive and highly recommended. Within such class of methods the most frequently used are the finite element method (FEM), (Zienckievicz and Morgan, 1983), and the finite volume method (FVM), (Barth, 1992). The later is particularly attractive in reservoir problems due to, among other things, its local and global conservation properties.

In this work the cell vertex finite volume formulation using median dual control volumes is implemented using an edge-based data structure that is adapted for solving two-dimensional convection-diffusion problems. In order to account for the non-viscous terms appearing in transport equations, a variation of the upwind scheme, adapted for use on unstructured meshes, is also utilized. This finite volume formulation is very flexible and efficient, and it is equivalent to the edge-based FEM when linear triangular elements are employed (Lyra, 1994; Sorensen, 2001). The formulation is flexible to deal with any kind of unstructured meshes with elements of different types. For instance, in 2-D either triangular, quadrilateral or mixed meshes can be directly used, and the same happens when dealing with 3-D, where tetrahedral, hexahedral, pyramids, prisms and mixed meshes can be adopted. In terms of efficiency, both memory and CPU time requirements are reduced by using an edge-based implementation (Barth, 1992; Sorensen, 2001). Finally, an edge-based data structure allows for the implementation of different types of finite difference discretization in the context of 2-D and 3-D unstructured meshes (Lyra, 1994).

2. Mathematical Model

In order to simplify our notation, but without loss of generality we present here the mathematical governing equations for immiscible biphasic flows of water and oil through rigid porous media. This model (which can be directly extend to miscible, three phase flow) is obtained combining the Darcy's Law with the mass conservation equation for each phase. The model adopted here has been successfully used by many authors (Peaceman, 1977; Ewing, 1983), though it is not commonly used in commercial reservoir simulators. Even though the approach to be used in our work seems more complex, it is far more useful when one aims for numerical accuracy and efficiency (Peaceman, 1977; Ewing, 1983; Da Silva, 2000).

The basic system of equations, given in Equations (1), (2) and (3), that arises from the combination of Darcy's Law and mass conservation equations for each phase are:

$$\phi c_T \frac{\partial p_m}{\partial t} + \nabla \mathbf{v_T} = Q_T \qquad (1)$$

$$\mathbf{v_T} = -\mathbf{\Lambda_1} \nabla p_m - \mathbf{\Lambda_2} \frac{dp_{cp}}{ds_w} \nabla s_w + (\rho_o \lambda_o + \rho_w \lambda_w)\mathbf{Kg} \qquad (2)$$

$$\phi \frac{\partial s_w}{\partial t} + \nabla \cdot \mathbf{v_a} + \nabla \cdot [\mathbf{D} \nabla s_w] + Q_w = 0 \qquad (3)$$

Where, $s_o + s_w = \mathbf{1}$, $\mathbf{v_T} = \mathbf{v_o} + \mathbf{v_w}$, $\mathbf{\Lambda_1} = \mathbf{K}(\lambda_o + \lambda_w)$ and $\mathbf{\Lambda_2} = \frac{1}{2}\mathbf{K}(\lambda_o - \lambda_w)$.

Equation 1 is the so called pressure equation and Equation 3 is the saturation equation. In these equations, p_m is the mean pressure and s_i with $i = w,o$ (water/oil) is the phase saturation and v_T (Equation 2) stands for the total velocity, which accounts for the coupling between Equations 1 and 3. For a complete description of the variables and the problem itself we strongly recommend Ewing, (1983). It is interesting to note that the pressure equation is a typical diffusion type equation (parabolic when solving transient problems and elliptic in a steady-state situation) while the saturation equation is a convection-diffusion type equation, being hyperbolic by nature.

3. Numerical Formulation

The numerical formulation used to discretize the previous equations is an edge based, cell vertex variation of the well known Finite Volume Method (FVM), (Sorensen, 2001; Lyra et al, 2002. In this kind of formulation, most of the coefficients necessary to our computation are associated with the edges of the mesh. The edge based finite volume formulation adopted is highly computationally efficient in terms of CPU time and memory use. Besides, this kind of approach allows the utilization of 1-D "upwind" type discretization techniques to be extended to 2-D and 3-D unstructured meshes without too many mathematical considerations (Lyra, 1994). This last point can be extremely useful due to the strong hyperbolic characteristic of some reservoir problems.

Nowadays there are many finite volume schemes that can be successfully used to solve balance equations. In this work we adopted a node centered median dual finite volume technique. In contrast to a cell centered formulation, in a node centered approach, the values of the unknowns are defined on the nodes and the control volumes (CV) are defined in a way that each mesh point is associated with only one CV.

In order to discretize the bidimensional domain, we have used triangular elements, though there is, in principle, no restriction to the shape of the elements that can be used in unstructured finite volume formulations. For a triangular mesh, the control volume cells were built connecting the centroids to the middle point of the triangles that surround a specific node. The control volume created in this fashion (known as "median dual") is quite general and the Voronoi diagrams are nothing but a special case of this scheme. The structure formed by the surfaces connecting the control volumes is denominated the dual mesh. Figure 1 shows the triangular mesh (red) utilized to solve the heat conduction problem of Section 4 and its correspondent dual mesh (blue). In node centered schemes, the fluxes are integrated on the dual mesh usually through a loop over the edges (2-D) or faces (3-D) and the computational cost is therefore, proportional to the number of edges or faces of the mesh.

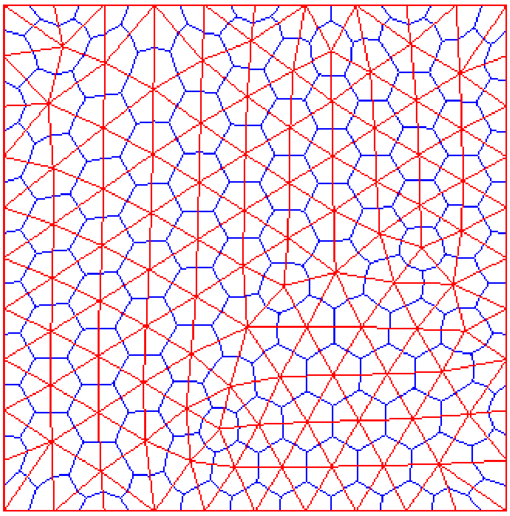

Figure 1. Unstructured triangular mesh (red) and its correspondent dual mesh (blue).

It is well known that central difference type methods such as the Galerkin method produce unstable numerical schemes when used to discretize the non-viscous (convective) terms that characterize transport equations (e.g. saturation equation). In order to overcome this difficulty we are testing different schemes to correctly treat convective terms. In this work we experimented a method that was originally proposed by Jameson et al (1981) with the modifications introduced by Peraire et al (1993). This method is based on the introduction of an adaptive artificial dissipative term that combines second order with fourth order diffusive terms (Lyra, 1994). The basic idea of the method is to introduce the second order terms in regions of high gradients and to use the fourth order terms only in regions of smooth gradients in order to stabilize the scheme.

4. Examples

In this section we show two numerical experiments using as model equations the linear heat conduction (diffusion type) and pollutant transport equations (convection-diffusion type) that respectively, resemble the pressure and the saturation equations that are typical of reservoir simulation problems.

The first one refers to a non-dimensional form of a steady-state, linear heat conduction problem over a square of constant thickness. We introduced a constant source term at the bottom left corner and a constant sink term at the upper right corner of the square similar to the well known quarter of five spot scheme (Ewing, 1983). The boundary conditions for this problem are null heat fluxes through the four lateral faces. For a non-dimensional form of the equations, the heat capacity is C_p = 1.0, the material conductivity is K = 1.0, the source term is Q_1= 100.0 and the sink term is Q_2 = -100.0. Figures 2 and 3 show respectively, the contours of temperature and the temperature distribution along the diagonal connecting the bottom left and upper right corners.

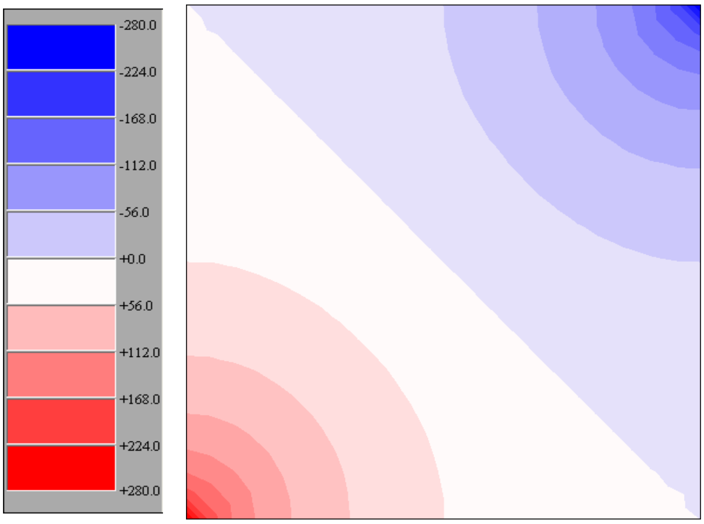

Figure 2. Contours of temperature for the problem of the heat-conduction in a square domain.

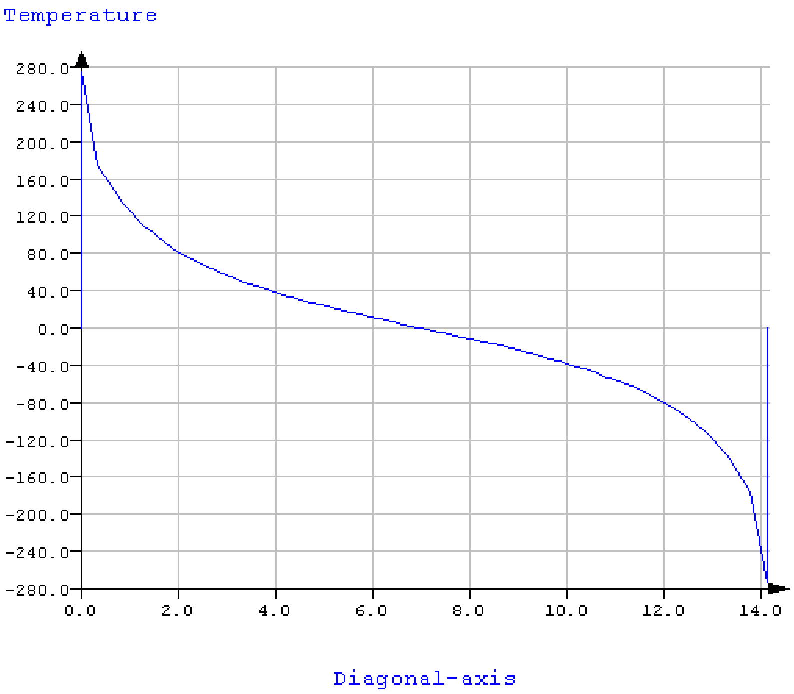

Figure 3. Temperature distribution along the diagonal connecting the source and sink corners, for the problem of heat conduction in a square domain.

The second example to be handled is the non-dimensional form of the transport of pollutants equation with a Peclet number Pe = 15.0 ($Pe = V.\Delta x / D$) in the x direction. The concentration of a substance that will be transported along the bi-dimensional domain is $C_0 = 10.0$ at $x = 0.0$, the velocity of transport is $V_x = 1.0$ while $V_y = 0.0$. The diffusion coefficients are $D_x = D_y = D = 0.12$. Figures 3 and 4 show, respectively, the contours of concentration and the concentration along x-axis ($y = 0.0$) at time $t = 50.0$.

As stated before, these equations are, respectively, simplified linear versions of the pressure and the saturation equations which we intend to cope with at the next phase of the work. In order to prevent or at least reduce oscillations in the solution, we used a “Switched Artificial Viscosity Approach” (Peraire et al, 1993), which is strongly affected by user defined parameters and which, in this example, was not capable of completely eliminating oscillations in the concentration front. Other alternatives to deal with the convective term will be investigated in order to improve the accuracy of the capture of the concentration front.

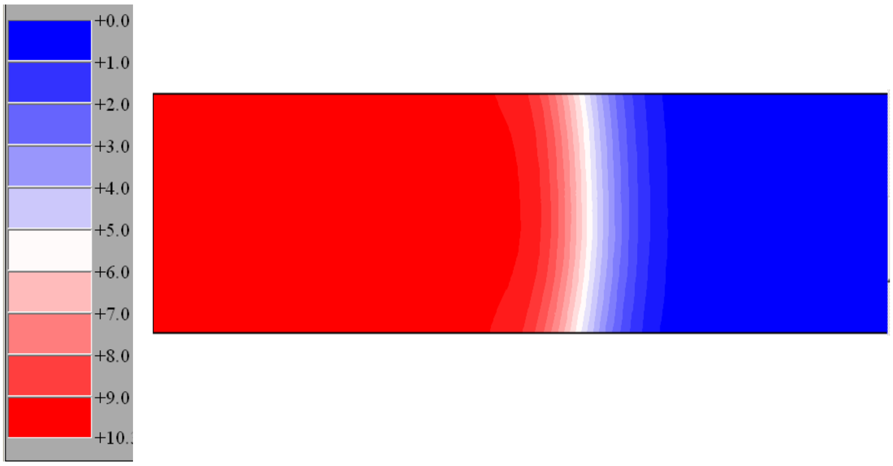

Figure 4. Contours of concentration for the problem of the transport of pollutants.

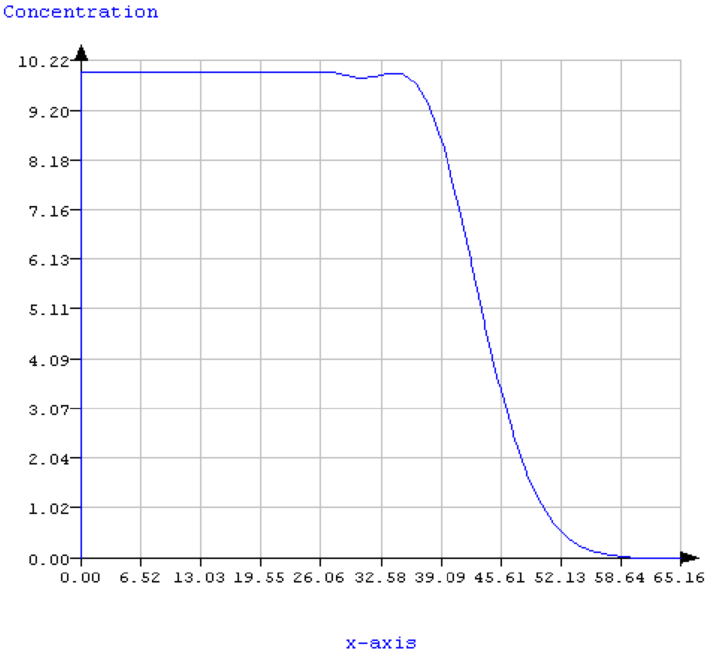

Figure 5. Concentration along x-axis ($y = 0.0$) for the problem of the transport of pollutants.

5. Further Issues

The next steps of our work include the use of the finite volume formulation to solve Equations 1 to 3 (oil–water problem) and later the multiphase flow model, and to exploit certain computational techniques, such as mesh adaptation, acceleration multigrid and the use parallel computing in distributed memory machines (cluster of Pcs).

5.1 Mesh Adaptation

The capability of automatic mesh adaptation can be of utmost importance in certain reservoir problems (Lazarov and Tomov, 2000), especially near production and injection wells, and on another regions of high gradients of saturation, which move on during the simulation time due to the complex physical characteristics of the multiphase flow through porous media. In order to correctly deal with these problems (at least in 2-D), we intend to adopt local and global remeshing procedures in which the mesh is rebuilt without increasing substantially the number of unknowns. (Lyra and Carvalho, 2000).

5.2 Multigrid Acceleration

Nowadays the multigrid method is considered one of the most important iterative methods to increase convergence rates when solving system of equations (Hirsch, 1988; Wesseling, 1991). This method has its origins in the properties of the classical iterative methods. Particularly, some of these methods have low convergence rates due to the low attenuation of low frequency errors. The basic idea of acceleration multigrid is to apply initially one or more steps of an iterative method with good smoothing properties of the high frequency components of the error and then transfer the problem to a coarser mesh, where an approximation to the solution correction is obtained with a lower computational cost due to the less degrees of freedom of the coarser mesh. Finally, the correction in the solution is transferred to the finer mesh in order to generate the new approximation to the solution. The utilization of this powerful iterative technique when solving the non-linear partial differential equations typically found in numerical reservoir problems is quite recent, being a promising field for future research.

5.3 Parallel Computing

The use of parallel computers (clusters of processors capable of working cooperatively in order to solve computational problems) enables the solution of problems with sizes almost impossible to be treated by computers with only one CPU. For distributed memory machines, the size of the problems that can be handled by the computer (e.g. clusters of personal computers) grows with the size of the cluster, by allowing each node of the computer to deal with a smaller part of the problem. It is important to note that with the appropriate programming technique the elapsed time used to solve a particular problem can be notably reduced. Most of the practical problems involving multiphase flow through porous media, and, especially those dealing with numerical reservoir problems have an extremely complex nature, requiring a huge computational effort to perform large scale simulations. Besides, they have many length scales that introduce further computational effort to correctly account for these characteristics. The use of parallel computing is, therefore, an important tool to be used in the context of the numerical reservoir simulation (Neyval et al, 2001).

6. Concluding Remarks

In this article we have made certain considerations about the mathematical aspects of the PDEs that describe the two phase flow in hydrocarbons reservoirs and we made some comments about the methods that will be used to solve them numerically. As a first step towards a numerical tool to solve multiphase reservoir problems, we have developed a computational code which is capable of solving some simple model examples that present the basic characteristics of the reservoir equations. Besides, certain subjects such as mesh adaptation, multigrid methods and parallelization, which are tools that will be explored more deeply in the near future, were briefly discussed. As a next step of this work, we expect to deal with more realistic problems such as the non-linear immiscible displacement model (oil-water).

7. Acknowledgments

The authors would like to acknowledge the Brazilian Research Council (CNPq) and the National Petroleum Agency (ANP) for their financial support provided during the development of this research.

8. References

BARTH, T.J. Aspects of unstructured grids and finite-volume solvers for the Euler and Navier-Stokes equations, AGARD Report 787, pp. 6.1-6.61, 1992.

DA SILVA, A. S. Métodos Adaptativos para a Simulação de Fluidos Bi-Fásicos em Meios Porosos, Rio de Janeiro, UFRJ, PhD Thesis, March, 2000 (in portuguese).

EWING, R. E. In: The Mathematics of Reservoir Simulation, Siam, Philadelphia, 1983.

HIRSCH, C. In: Numerical Computation of Internal and External Flows, Vol.1., John Willey & Sons, U.K., 1988.

JAMESON, A., SHMIDT, W., TURKEL, E. Numerical Simulation of the Euler Equations by Finite Volume Methods Using Runge-Kutta Time Stepping Schemes, Technical report 81-1259, AIAA Paper, 1981.

LAZAROV R. D., TOMOV, S. Z. Adaptive Finite Volume Element Method For Convection-Diffusion-Reaction Problems in 3-D, Proc. Int. Symposium on Large Scale Scientific Computing, Kananaskis, Canada, 2000.

LYRA, P. R. M. Unstructured Grid Algorithms for Fluid Dynamics and Heat Conduction, Ph.D. thesis C/PH/182/94, Department of Civil Engineering – University of Wales/Swansea- UK , 1994.

LYRA, P. R. M., FERNANDES DE LIMA, R. C., GUIMARÃES, C. S. C., DE CARVALHO, D. K. E. An Edge-Based Unstructured Finite Volume Method for the Solution of Potential Problems, Proc. MECOM'2002 - First South American Congress on Computational Mechanics, Parana and Santa Fé, Argentina, November, 2002, pp. 1-19, in CD-ROM.

NEYVAL, C. R. J., DE ANGELI, J. P., DE SOUZA, A. F., LOPES, R. H. C. Petroleum Reservoir Simulation Using Finite Volume Method with Non-Structured Grids and Parallel Distributed Computing, 22nd CILANCE, Campinas, Brasil, November, 2001, in CD-ROM.

PEACEMAN, D. W. In: Fundamentals of Reservoir Simulation, Elsevier Scientific Publishinhg Company, Amsterdam, 1977.

PERAIRE, J., PEIRÓ, J., MORGAN, K. Finite element multigrid solution of euler flows past installed aero-engines, V.11.433-451, *J. Computational Mechanics*, 1993.

SORENSEN, K. A. A Multigrid Accelerated Procedure for the Solution of Compressible Fluid Flows on Unstructured Meshes, Swansea, PhD Thesis, December, 2001.

WESSLING P., An introduction to Multigrid Methods, John Wiley & Sons, Univ. of Technology, Netherlands, 1991.

ZIENKIEWICZ, O. C., MORGAN, K. *Finite element and approximation*, John Wiley & Sons, Inc., 1983.

LYRA, P.R.M., CARVALHO, D.K.E. A Flexible Unstructured Mesh Generator for Transient Anisotropic Remeshing, European Cong. on Comp. Meth. in Appl. Scienc. and Eng. (ECCOMAS), Barcelona-Spain, September, 2000.