COBEM2015

ABCM Associação Brasileira de Engenharia e Ciências Mecânicas

23rd ABCM International Congress of Mechanical Engineering
December 6-11, 2015, Rio de Janeiro, RJ, Brazil

RESERVOIR SIMULATION MODELING IN CFD ENVIRONMENT

R. A. C. Dias, A. O. Nieckele

Dept. Eng. Mecânica, PUC-Rio, R. Marques de S. Vicente 225, Rio de Janeiro, RJ, 22451-900, Br

diascrodrigo@gmail.com, nieckele@puc-rio.br

P. D. Fernandes, S. D. Souza, T.J.L. Oliveira

Petrobras CENPES, Av. Horácio Macedo, 950, Prédio 20, Sala 2040, Cidade Universitária, Rio de Janeiro, RJ, Br

pdore@petrobras.com.br, sylviods@petrobras.com.br, tjudson@petrobras.com.br

A. O. S. Moraes, J. L. Favero

Wikki Brasil C. E. LTDA. Rua Hélio de Almeida s/n, sala 31. Cidade Universitária, Rio de Janeiro, RJ, 21.941-614, Br

antonio.samel@wikki.com.br, jovani.favero@wikki.com.br

Abstract. *The accurate computation of well performance is very important for reservoir management and optimization. Commercial standard large-scale reservoir simulators, such as IMEX or Eclipse, do not consider important near-well physics such as well completion components, details around the wellbore and adequate reservoir heterogeneity since these effects occur on scales that are much smaller than the reservoir scale. However, the exact wellbore geometry, perforations and hydraulic fractures are examples of near well details that can be accounted in a simple way into CFD software's. On the other hand, CFD software's do not have a mathematical model of the hydraulic diffusivity, which is extremely widespread in the petroleum industry. Aiming to predict the flow through the reservoir as well as at the wellbore, at the present paper a hydraulic diffusivity mathematical model was implemented in the open source code OpenFOAM and in ANSYS Fluent through UDFs macros. The solutions obtained employing both CFD codes, were compared with the predictions of the well-established commercial reservoir simulation software CMG IMEX, presenting excellent agreement. It was also shown that superior and more detailed solution can be achieved.*

Keywords: *Reservoir Simulation, CFD, Near Well, OpenFOAM, Fluent*

1. INTRODUCTION

Reservoir simulation is the process of determining numerically the flow field in a petroleum reservoir system through the use of mathematical models that represent the fluid physical behavior. Reservoir simulation models are used by petroleum companies to help in the development of new fields and in developed fields where production forecasts are needed to help make investment decisions. It is an important tool that facilitates reservoir engineers as they work to optimize the design of well development schemes, improve the efficiency of reservoir development, and enhance oil and gas recovery.

The quality of reservoir predictions depend on a number of factors, such as raw data measurements, data processing, data interpretation and petrophysical modeling. The most important factor is an accurate and detailed description of the reservoir, which are necessary to defined the boundary conditions for the flow equations in a porous rock, and those are known only with a great deal of uncertainty.

Nevertheless, reservoir simulation is regularly used with great success, and remains one of the support pillars for decision making when field plans for development and operations are made. Even if there may be uncertainties tied to simulated results, the simulations reveal invaluable information about the reservoir flow, and not the least, can point to areas which need closer investigation or can represent elements of risk to the production scenarios.

Commercial reservoir simulators such as IMEX and Eclipse has the ability to predict the performance of a petroleum reservoir taking into account a large number of wells. These softwares are able to determine the possible production rates

and overall production resulting from different production strategies. Standard large-scale reservoir simulators, however, typically do not consider important near-well physics such as well completion components and details around the wellbore. Depending on the type of problem, when these small scales effects can play a significant role, theses standards softwares are not be able to provide good results.

CFD commercial simulators have several models implemented in their solvers for particular situations. Turbulence, multiphase flow, combustion are examples of physical models widespread in computational fluid dynamics area. For each physical phenomena mentioned, there are several models to predict the flow behavior. However, CFD commercial simulators do not have a fluid flow model adapted for the flow behavior in petroleum reservoir. All capacity and versatility to geometry and mesh generation, all possibilities of numerical methods and nonlinear systems solution can be used with great potential for reservoir simulation if specific models were available into CFD softwares.

This paper presents the results of a single-phase compressible flow in petroleum reservoir considering a Black Oil fluid. To obtain the solution the hydraulic diffusivity equation was implemented in the open source code OpenFOAM and in ANSYS Fluent through UDF (User Defined Functions) macros. This model is fairly common and well known in commercial software for reservoir simulations, such as IMEX or Eclipse. The solutions obtained for a few cases with the present methodology, employing both CFD codes, were compared with the predictions of the well established commercial reservoir simulation software CMG IMEX, presenting excellent agreement.

2. MATHEMATICAL MODEL

The flow through a porous media can be modeled by at least two approaches. The local volume-averaging technique and mixture theory. Macroscopic equations obtained after volume integration over a Representative Elementary Volume (REV) were discussed in detail by Whitaker (1966); Quintard and Whitaker (1994), to describe quantities such as temperature, pressure, concentration, and the velocity components. Mixture theory allows a local description of the flow in a porous medium, supported by a thermodynamically consistent theory which generalizes the classical continuum mechanics discussed in detail by Green and Naghid (1965), Atkin and Craine (1976) and others.

These equations are transient non-linear equations, with high cost to obtain a numerical solution. As an alternative, most reservoir simulation are performed with Darcy's law, which is a phenomenologically law that describes the flow of a fluid through a porous medium. Although Darcy's law was determined experimentally by Darcy, it can be derived from the Navier-Stokes equations by local volume-averaging technique and also by mixture theory (Craft and Hawkins, 1991). Recently Dias *et al.* (2012) presented a numerical comparison between mixture theory and the formulation based on Darcy and showed that in reservoir conditions both formulations lead to same results. The Darcy's law can be written as:

$$\mathbf{v} = -\frac{\mathbf{K}}{\mu} \cdot (\nabla p - \rho\, \mathbf{g}) \tag{1}$$

At the present work, the Black Oil model, usually employed by the petroleum industry, was selected to determine the fluid properties. Considering an isothermal flow, the fluid density in reservoir conditions depends on the oil density measured at standard conditions ρ^{sc}, the ratio of gas solubility in the oil R_S, the gas density in standard conditions ρ_g^{sc} and the oil formation volume factor B_o, which depends on pressure.

$$\rho(p) = \frac{\rho^{sc} + \rho_g^{sc} R_S}{B_o(p)} \qquad \text{and} \qquad B_o(p) = B_{ob}\left[1 - c_f\,(p - p_b)\right] \qquad ; \qquad c_f = \frac{1}{\rho}\frac{\partial \rho}{\partial p} \tag{2}$$

where B_{ob} is oil volume factor at bubble point, p_b is the bubble point, and c_f is the fluid compressibility.

In the Black Oil model, the fluid viscosity as well as rock porosity are also functions of pressure,

$$\mu(p) = \mu_b + C_{visc}\,(p - p_b) \qquad ; \qquad \phi(p) = \phi^{ref}\left[1 + c_r\,(p - p_{ref})\right] \qquad ; \qquad c_r = \frac{1}{\phi}\frac{\partial \phi}{\partial p} \tag{3}$$

where μ_b is fluid viscosity at bubble point, C_{visc} is proportional coefficient for viscosity, ϕ^{ref} is porosity at reference

pressure and p_{ref} is reference pressure, and c_r is the rock compressibility.

The Darcy's Law can be combined with the mass conservation law $\partial(\phi\rho)/\partial t + \nabla \cdot (\rho \mathbf{v}) = 0$ to obtain an hydraulic diffusivity equation,

$$\frac{\partial(\phi\rho)}{\partial t} + \nabla \cdot \left(-\rho \frac{\mathbf{K}(\mathbf{x})}{\mu(p)} \cdot (\nabla p - \rho\, \mathbf{g}) \right) = 0 \tag{4}$$

which represents the behavior of a single phase flow of compressible liquids in compressible porous media. The dependence of pressure in density and porosity can be introduced in the equation in order to develop a general equation for pressure evolution, by $\rho = \psi p$ and $\phi = \Phi p$. Where ψ and Φ can be general state functions for pressure, temperature, fluid composition and rock formation. In the present study these functions only depend on pressure. The transient term of Eq. (4) can be rewritten as $\Gamma(p)\partial p/\partial t$, where the Γ function is

$$\Gamma(p) = \rho \left(\Phi + p \frac{\partial \Phi}{\partial p} \right) + \phi \left(\psi + p \frac{\partial \psi}{\partial p} \right) = \rho\phi \left(c_r - \frac{1}{B_o} \frac{\partial B_o}{\partial p} \right) \tag{5}$$

Finally, Eq. (4) can be rewritten generically in terms of fluid pressure, where $\nu = \mu/\rho$, resulting in the hydraulic diffusivity equation

$$\Gamma(p) \frac{\partial p}{\partial t} = \nabla \cdot \left(\frac{\mathbf{K}}{\nu(p)} \cdot \nabla p \right) - \nabla \cdot \left(\frac{\rho(p)}{\nu(p)} \mathbf{K} \cdot \mathbf{g} \right) \tag{6}$$

3. NUMERICAL MODEL

Finite volume method (FVM) uses a volume integral formulation of the problem with a finite partitioning set of volumes to discretize the equations. The FVM is the main method implemented in several dedicated softwares such as ANSYS CFX, ANSYS Fluent, StarCCM, CFD++ and OpenFOAM, the latter being open source. The main characteristic of the FVM, which is responsible for its disseminated use in CFD methods, is that it guarantees global conservation of the variables of interest. The method can be applied to generalized meshes, i.e., it does not impose any constraints to the structure of the mesh neither to the type or form of its elements. In other words, the method has support to structured or non structured polyhedral meshes (Jasak, 1996).

At the present paper, the hydraulic diffusivity equation adequate to reservoir simulation, presented at the previous section, was implemented in CFD softwares. The code uses classical finite volume method with co-located arrangement of variables in unstructured polyhedral meshes, thereby enhancing the ability to solve various near well problems such as wellbore, perforations and hydraulic fractures geometries. Methods are needed to discretization the transient, diffusive, gradient and divergent operators, as detailed in the following sections.

3.1 Transient Term

For the transient term, the volume integral was approximated by the mean value theorem, resulting in

$$\int_{\forall_P} \Gamma \frac{\partial p}{\partial t} d\forall \cong \Gamma_P \left(\frac{\partial p}{\partial t} \right)_P \forall_P \tag{7}$$

where the subscript P is the centroid index of the mesh element over which the term is integrated and $\forall_P$ is their respective volume. The pressure time derivative in the P element can be evaluated by different schemes, such as Euler (1th order accuracy) and backward (2nd order accuracy) formulas given, respectively by

$$\left(\frac{\partial p}{\partial t} \right)_P \cong \frac{\left(p_P^{n+1} - p_P^n \right)}{\Delta t} \quad ; \quad \left(\frac{\partial p}{\partial t} \right)_P \cong \frac{\left(3p_P^{n+1} - 4p_P^n + 3p_P^{n-1} \right)}{2\Delta t} \tag{8}$$

where Δt is the time step size, n is the index of the current time t, thus $n+1$ corresponds to the pressure at $t+\Delta t$ time, and $n-1$ refers to the pressure at the previous time, corresponding to the time $t-\Delta t$ for fixed time step size.

3.2 Gradient Evaluation

The velocity field calculation, and consequently the flow rate at wellbore boundary condition, must be know and are determined explicitly at end of each time step. Therefore, since the velocity is directly proportional to pressure gradient, its evaluation is crucial to obtain accurate results.

Gradient at the centroid P can be calculated in different ways, such as (i) the Gauss integration and (ii) the Least Square Method. These two methods are available in OpenFOAM and Fluent softwares, which also provide the gradient evaluation at the control volume faces.

The first possibility is use Gauss theorem and applying the mean value theorem, the discretized gradient at the centroid point P is given by

$$(\nabla p)_P \cong \frac{1}{\forall_P}\sum_f \mathbf{S}_f p_f \tag{9}$$

The second possibility to evaluate the gradient is the Least Square Method is given by Eq. (10).

$$(\nabla \phi)_P = \sum_N w_N^2 \mathbf{G}^{-1}\mathbf{d}(\phi_N - \phi_P) \qquad \text{and} \qquad w_N = \frac{1}{\|\mathbf{d}\|} \tag{10}$$

where N extends to all neighbors of the cell P, $\mathbf{d}$ is the vector from centroid P to the centroid N and the weights w_N and the matrix $\mathbf{G}$ are given by Eq. (11):

$$\mathbf{G} = \sum_N w_N^2 \mathbf{d}_N \otimes \mathbf{d}_N \tag{11}$$

It is important to mentioned that the least square method does not suffer with skewed meshes, preserving second order accuracy even for highly non-orthogonal ones. For orthogonal meshes, however, both methods have the same accuracy and in this situation, the Gaussian integration is preferred because it is computationally cheaper. For near well simulations is advisable to use the least squares method for the reasons described above.

3.3 Diffusive Term

The Gauss divergence theorem is also used for the discretization of the diffusive term. Applying the mean value theorem for the volume integral, we obtain

$$\int_{\forall_P} \nabla \cdot \left[\left(\frac{\mathbf{K}}{\nu}\right)\cdot \nabla p\right] d\forall = \oint_{\partial \forall_P} [\mathbf{n}\cdot(\tilde{\mathbf{K}}\cdot\nabla p)]dS \cong \sum_f \mathbf{S}_f \cdot (\tilde{\mathbf{K}}\cdot\nabla p)_f \qquad ; \qquad \tilde{\mathbf{K}} \equiv \frac{\mathbf{K}}{\nu} \tag{12}$$

To evaluate the scalar product between the face normal vector and the vector $\tilde{\mathbf{K}}\cdot\nabla p$ at the faces of non-orthogonal meshes, the permeability tensor ($\tilde{\mathbf{K}}$) is decomposed into an isotropic and deviatoric component according to Eq. (13):

$$\tilde{\mathbf{K}} = \gamma\,\mathbf{I} + \mathbf{D_k} \qquad \text{where} \qquad \gamma = \frac{tr\tilde{\mathbf{K}}}{3} \qquad \text{and} \qquad \mathbf{D_k} \equiv \left[\tilde{\mathbf{K}} - \gamma\mathbf{I}\right] \tag{13}$$

Therefore, the sum of the right side of Eq. (12) can be rewritten by:

$$\sum_f \mathbf{S}_f \cdot \left(\tilde{\mathbf{K}}\cdot\boldsymbol{\nabla} p\right)_f = \sum_f \gamma_f \left(\mathbf{S}\cdot\boldsymbol{\nabla} p\right)_f + \left[\sum_f \mathbf{S}_f \cdot \left(\mathbf{D_k}\cdot\boldsymbol{\nabla} p\right)_f\right]^* \tag{14}$$

where the index superscript $(*)$ indicates an explicit evaluation.

The treatment of the first term on the right hand side of Eq. (14) was made by following the procedure for non-orthogonality correction described in Moraes *et al.* (2013) and Jasak (1996). The vector $\mathbf{S}_f$ is divided in two vectors, a vector aligned with $\overline{PN}$, define as $\mathbf{\Delta}$, and another one, corresponding to the difference between the vectors $\mathbf{S}_f$ and $\mathbf{\Delta}$, which is defined as $\mathbf{k}$. Equation (15) shows the application of this division of vectors in first term on right side of Eq. (14).

$$(\boldsymbol{\nabla} p)_f \cdot \mathbf{S}_f \cong (\boldsymbol{\nabla} p)_f \cdot \mathbf{\Delta} + [(\boldsymbol{\nabla} p)_f \cdot \mathbf{k}]^* \tag{15}$$

where the vector $\mathbf{\Delta}$ and $\mathbf{k}$ are defined according to the Eq. (16) (Jasak, 1996).

$$\mathbf{\Delta} = \frac{\mathbf{d}}{\mathbf{d} \cdot \mathbf{S}_f} |\mathbf{S}_f|^2 \qquad \text{and} \qquad \mathbf{k} = \mathbf{\Delta} - \mathbf{S}_f \tag{16}$$

The first term on the right side of the Eq. (15) is implicitly evaluated using the neighboring cells to the face f as

$$\mathbf{\Delta} \cdot (\boldsymbol{\nabla} p)_f = |\mathbf{\Delta}| \frac{p_N - p_P}{|\mathbf{d}|} \tag{17}$$

The second term on the right hand side of the equation Eq. (15), i.e, the non-orthogonal correction, and the second term on the right hand side of the equation Eq. (14), the portion associated with deviatoric part of the tensor $\tilde{\mathbf{K}}$, i.e, the anisotropy correction, are treated explicitly.

3.4 Source Term

Since the source term of hydraulic diffusivity equation does not depend explicitly on pressure, their evaluation is done explicitly, using the Gauss divergence theorem and the mean value theorem, as show in Eq. (18).

$$\int_{\forall_P} \boldsymbol{\nabla} \cdot \left[\frac{\rho_o}{\nu} \mathbf{K} \cdot \mathbf{g} \right] d\forall = \oint_{\partial \forall_P} \frac{\rho_o}{\nu} (\mathbf{n} \cdot \mathbf{K} \cdot \mathbf{g}) \, dS \cong \sum_f \left[\frac{\rho_o}{\nu} (\mathbf{S} \cdot \mathbf{K} \cdot \mathbf{g}) \right]_f \tag{18}$$

Thus, the source term generates a contribution to the independent vector of the pressure linear system. The terms on the faces can be obtained by interpolating the values in element centroids. In case where the contribution of this term is very relevant and highly anisotropic mesh is used, the skewness correction should be used to ensure 2^{nd} order accuracy in solution.

4. RESULTS

In order to evaluate the reservoir code developed within CFD softwares (i.e. Ansys Fluent and OpenFOAM), a set of simple examples were proposed and compared with the reservoir simulations obtained with the commercial software CMG-IMEX. This software is a three-phase black oil simulator software, and it is widely used by the oil industry, therefore, it was chosen to validate the implementation developed. Three different test cases are presented to validate the developed implementation in the CFD softwares: (i) homogeneous reservoir with constant properties, (ii) homogeneous reservoir with variable properties (black oil model) and (iii) heterogeneous reservoir with variable properties (black oil model).

For the presentation of the results, the solution obtained with the model implemented in OpenFOAM is referred to as *hydraulicDiffusivityFoam*. Table 1 shows the values for homogeneous reservoir with constant properties data (case A) and homogeneous reservoir with variable properties data (case B).

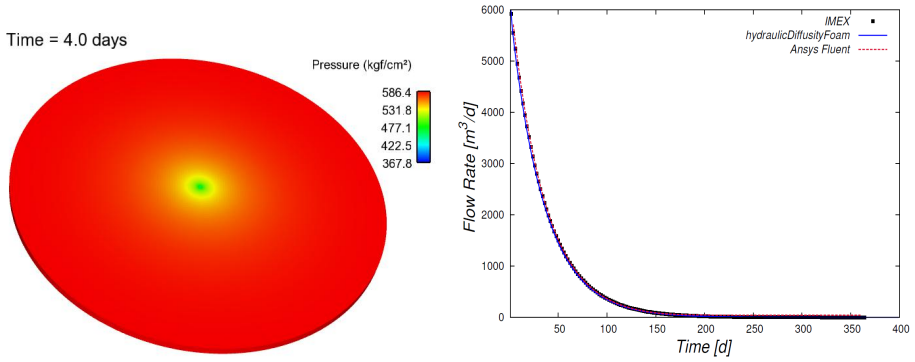
Table 1. Homogeneous reservoir with constant and variable properties data.

Fluid Properties				Formation Properties			
Properties	Dimension	Case A	Case B	Properties	Dimension	Case A	Case B
μ_b	[Kg/(m s)]	$1.0 \cdot 10^{-3}$	$1.0761 \cdot 10^{-3}$	$K_{xx}; K_{yy}; K_{zz}$	[m^2]	$1.0 \cdot 10^{-13}$	$9.87 \cdot 10^{-14}$
C_{visc}	[ ]	0	$5.4473 \cdot 10^{-12}$	s_{wcon}	[ ]	0	0
p_b	[Pa]	$308.26 \cdot 10^5$	$308.26 \cdot 10^5$	p_{ref}	[Pa]	$627.6256 \cdot 10^5$	$549.17 \cdot 10^5$
ρ_o	[Kg/m^3]	897.1239	897.1239	ϕ_{ref}	[ ]	$0,1$	$0,1$
R_s	[ ]	0	0	c_f	[m s^2/Kg]	$7.25 \cdot 10^{-10}$	$7.25 \cdot 10^{-10}$
c_o	[m s^2/Kg]	0	$7.25 \cdot 10^{-10}$				
B_{ob}	[ ]	1.0	1.4389				

4.1 Homogeneus Reservoir - Constant Propreties

The first case chosen is the simplest reservoir model. The reservoir is circular with fluid and formation properties constants and homogeneous permeability field. The results obtained with the solvers implement inside OpenFOAM and Ansys Fluent softwares, can be seen in Fig. 1(a). For this case the chosen software for pressure field post processing was Ansys Fluent. The pressure distribution after four days is shown. The pressure distribution is symmetric and significant smaller at the well as expected.

The most important information in a reservoir simulation is wellbore production, which is shown in Figure 1(b), where a comparison of the soltuion of the well flow rate over time obtained with the three softwares between can be seen. Good results were found when comparing OpenFoan, Fluent and IMEX softwares.



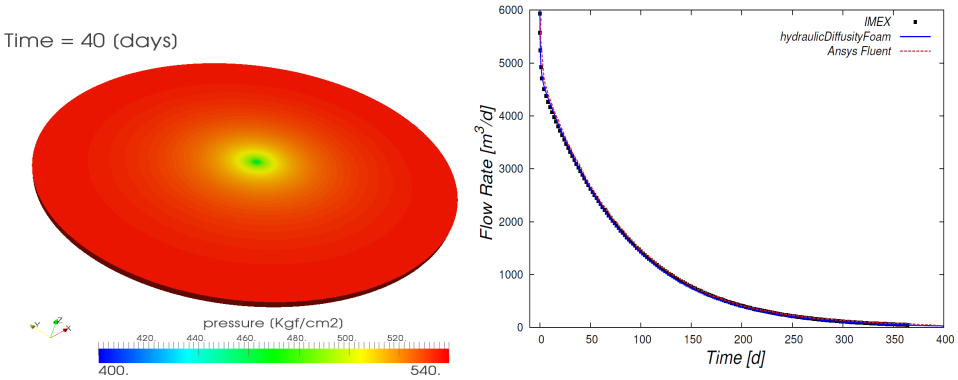
(a) Pressure field - Fluent results post processing (b) Comparison of wellbore rate

Figure 1. Results for homogeneous reservoir with constant properties.

4.2 Homogeneus Reservoir - Variable Propreties

This test case is aimed to validate the black oil model implementation. Therefore, for this case, the properties vary according with Eq. (2) and Eq. (3). The results for validation of black oil model into OpenFOAM and Ansys Fluent softwares can be seen in Fig. 2(a) and Fig. 2(b). The pressure field after 40 days of production is shown in Fig. 2(a). The influence of the nonuniform properties in the pressure field is negligible, since very similar results was obtained for the constant property case (Fig. 1(a)).

Once again, by examining Fig. 2(b), it can be seen that the same time evolution of the flow rate was obtained with the three softwares, indicating that the Black Oil Model was correctly treated.



(a) Pressure field - OpenFOAM results post processing

(b) Comparison of wellbore rate

Figure 2. Results for homogeneous reservoir and variable properties.

4.3 Heterogeneus Reservoir - Variable Propreties

The last test case corresponds to an heterogeneous reservoir. For this test, in order to allow a faithful comparison between the results obtained with the implementation performed in the CFD softwares with IMEX model, it was necessary to developed an algorithm by creating some classes in C ++ to transfer properties between the two softwares, in order to read a structured mesh in an IMEX input file. These classes map reservoir properties such as permeability and porosity between IMEX and CFD mesh elements. The scheme of properties transfer between IMEX and CFD mesh is shown in Fig. 3. The properties of the "Mesh 1" (mesh with 4 rectangles - representing IMEX mesh) are transferred to the "Mesh 2" (triangle mesh - representing CFD mesh).

Figure 4(a) shows the permeability distribution in the reservoir IMEX software. Figure 4(b) shows the permeability field exported from IMEX to OpenFOAM software, showing the the transfer of information was correctly achieved. In this test, the values for reservoir and fluid properties are identical to the previous case except for permeability which is imported from IMEX field and the value of the formation compressibility $c_f = 5.0985e^{-10}$ Pa^{-1}.

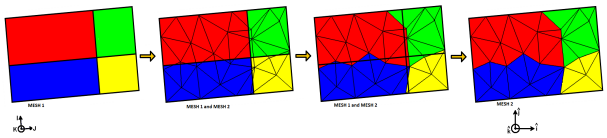
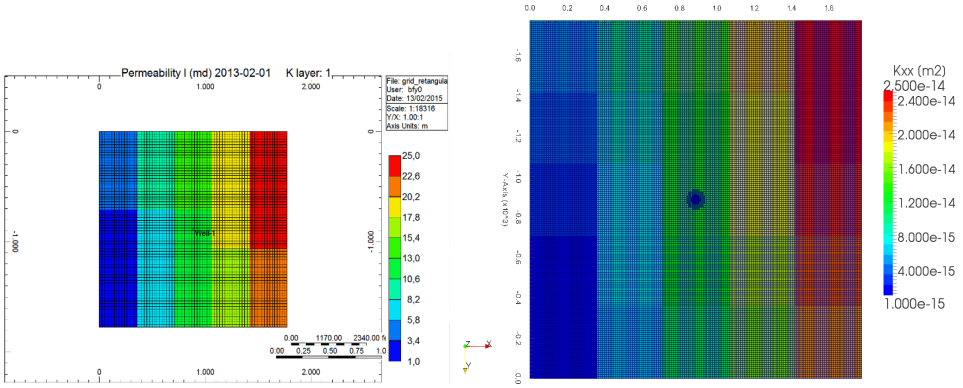


Figure 3. Properties transfer scheme between IMEX and Fluent mesh.

The results for validation of heterogeneous model into OpenFOAM and Ansys Fluent softwares can be seen in Fig. 5(a) and Fig. 5(b). Fig. 5(a) shows the pressure field along the reservoir after 400 days, where an asymmetry of the pressure distribution can be observed. The agreement between the time flow rate evolution shown in Fig. 5(b) indicate that the models implemented in both CFD solvers taking into account heterogeneous reservoir model are correct.

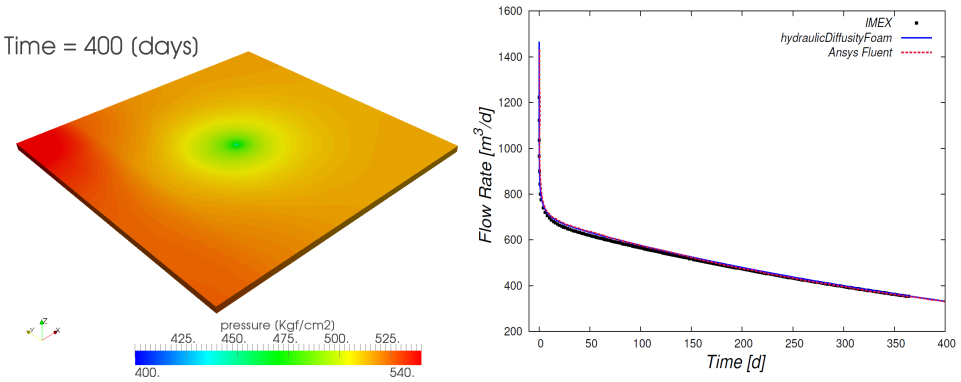
5. CONCLUSION

At the present work, a model to simulate the flow in a petroleum reservoir was implemented in two CFD codes (Fluent and OpenFoam). Both softwares were tested and their predictions were compared against the well established commercial software IMEX. Three test cases with increasing complexity were investigated, and perfect agreement was obtained among the softwares. These results indicate that the hydraulic diffusive equation necessary to simulation the flow through the reservoir is adequately implemented in both CFD codes. This will allow to perform a simultaneous investigation of the flow in the reservoir (large scale) and at the well region (small scale), which will be accomplished as a continuation of the present work.



(a) IMEX permeability field (b) CFD permeability field

Figure 4. Examples of permeability field exported from IMEX to OpenFOAM.



(a) Pressure field - OpenFOAM results post processing (b) Comparison of wellbore rate

Figure 5. Results for heterogeneous reservoir and variable properties.

6. ACKNOWLEDGEMENTS

The authors acknowledge support from CAPES, CNPq and Petrobras for continuous support.

7. REFERENCES

Atkin, R.J. and Craine, R.E., 1976. *Q. J. Mech. Appl. Math.*, Vol. XXIX, p. 209.

Craft, B. and Hawkins, M., 1991. *Applied Petroleum Reservoir Engineering*. Prentice Hall, 2nd edition.

Dias, R.A.C., de Oliveira, T.J.L. and Fernandes, P.D., 2012. "Numerical simulation of transient pressure data in porous media by mixture theory and darcy law". In *Proceedings of the ENCIT 2012*. Rio de Janeiro, RJ, Brazil.

Green, A.E. and Naghid, P.M., 1965. *Int. J. Engng. Sci.*, Vol. 3, p. 231.

Jasak, H., 1996. *Error Analysis and Estimation for the Finite Volume Method with Applications to Fluid Flows*. Ph.D. thesis, Imperial College of Science, Technology and Medicine, London.

Moraes, A.O.S., Cunha, C.G., Lage, P.L.C. and Silva, L.F.R., 2013. "Analysis of the non-orthogonality correction of finite volume discretization on unstructured meshes". In *COBEM 2013*. Ribeirão Preto, SP, Brazil.

Quintard, M. and Whitaker, S., 1994. "Transport in ordered and disordered porous media ii: Generalized volume averaging". *Transport in Porous Media*, Vol. 14, pp. 179–206.

Whitaker, S., 1966. "The equations of motion in porous media". *Chemical Engineering Science*, Vol. 21, pp. 291–300.

8. RESPONSIBILITY NOTICE

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