CFD - Recent efforts at the Computational Fluid Dynamics Laboratory - SINMEC/UFSC

Clovis R. Maliska

SINMEC
Mechanical Engineering Department - UFSC
P.O. Box 476
88040-900 - Florianópolis - SC - BRAZIL

ABSTRACT

The Computational Fluid Dynamics Laboratory (SINMEC) is devoted to the development of numerical methods for the solution of fluid flow and heat transfer problems encountered in several branches of engineering. The numerical technique uses boundary-fitted coordinate systems and the finite-volume concept to construct the discretized equations.

The present paper reports the recent developments in the area of Computational Fluid Dynamics applied to Aerodynamics, Petroleum Engineering and Environmental Flows. In addition it is also reported the efforts in developing unstructured finite volume methods for fluid flow problems and in developing a powerfull tridimensional scientific visualization packgage.

1. INTRODUCTION

Four main lines of research are under consideration. They are, the development and improvement of numerical techniques using structured(boundary-fitted) and unstructured grids, aerodynamics, petroleum engineering and environmental flows. A brief description of the models are presented with the estimation of the computer needs for their calculation.

2. RESEARCH ACTIVITIES

2.1. Aerodynamics

The interesting characteristic of the developed model is its ability in solving low as well as high Mach number flows [1,2,3], compared to the usual models available in the literature, where only one flow regime can be solved. This feature allows a broad application of the model in engineering problems. For example, aerodynamics of planes and vehicles, flow over buildings and hills, which are in the class of subsonic flows, and flows over rocktes, which are in the class of supersonic flows, can be handled using this model.

The mathematical model comprises the mass conservation, the three components of the momentum conservation and energy conservation equations, with the state equation for closing the equation system. The equation system is solved implicitly in a segregated manner, using the well known pressure-velocity coupling methods for transforming the mass conservation equation in an equation for pressure. So. six unknown with five partial differential equations and one algebraic equations need to be solved.

Typically. the size of the problems already solved corresponds to around 150.000 volumes, requiring about 6 CPU hours in a C-210 CONVEX machine. This amounts in solving five linear systems with 150.000 equations for each time step. without counting the iteration cicle inside each time step. Several complex three-dimensional flows. in the subsonic and supersonic regimes, were solved with extensive validation of the numerical results with the experimental ones. These results were extensively published. and are not reproduced here due to the lack of space. In this paper a numerical solutions of the incompressible flow over a sport vehicle is shown in Figure 1.

2.2. Petroleum Reservoir Simulation

The most recent 3D numerical model under development at the Laboratory aims the simulation of the multiphase flow in a porous medium. In the petroleum reservoir simulation this amounts in solving the flow of water, oil and gas in the reservoir. The model also uses boundary-fitted coordinates which helps in discretizing the irregular shapes of the reservoirs. The features of the model are now summarized; a) 3D arbitrary reservoir with variable porosity and permeability and two-phase flow of oil and water; b) injection and production wells with any completation; c) horizontal as well as vertical wells can be simulated; d)IMPES method for solving pressure and saturations; e) controlled time step according to the variation of pressure and saturations: f) wells can be opened and closed at any time: g) specification of geological faults inside the domain.

The method uses the concept of fault coefficients defined in the input data and boundary conditions such that only one equation is valid for interior and boundary control volumes. This simplifies considerably the routine for calculating the coefficients. Several bidimensional and tridimensional flows were solved sing the methodology[4,5]. As an ilustrative example, Figure 2 shows the saturation contours for the 3D simulation of a arbitrary reservoir. Typically the size of the mesh for this type of calculation contains 10.000 volumes. The CPU time to run the problem to a time correponding to 2.0 porous volume injected is around 30 minutes in a CONVEX C-210.

2.3. Enviromental flows

In this area several research projects are under development. Numerical models employing the similarity between compressible flows and open channels flows with the shallow water equations are been constructed. For more complex calculations 3D dimensional models for turbulent flow in water bodies and atmosfere are also in development. Figure 3 depicts the concentration of a pulse of a polutant discharged into a lake with mild water currents for six different times.

2.4. Unstructured grids

The use of unstructured grid is also a important topic of research at the Laboratory. The grid used is of the Voronoi type, since it has several attractive features for calculating fluxes at the control volume interfaces. This type of discretization is being used in petroleum reservoir simulations. Figure 4 shows a two-dimensional grid constructed with a grid generator developed at the Laboratory[6] for the simulation of the two-phase flow of oil and water[7]. This type of discretization is very promising for simulating fluid flow problems and extensions to 3D is under consideration.

2.5. Scientific visualization

A lot of efforts have been devoted to the development of the ISO-3D visualization package[8,9]. It is fully portable to any workstation, written in C++, running in a UNIX compatible operational system. The package is now a very powerful tool for CFD and other applications which involves tridimensional visualization. All figures shown in this paper were obtained using the ISO-3D software. The ISO-ANIMATOR routine is already connected to the ISO-3D package for animation purposes.

3. CONCLUSIONS

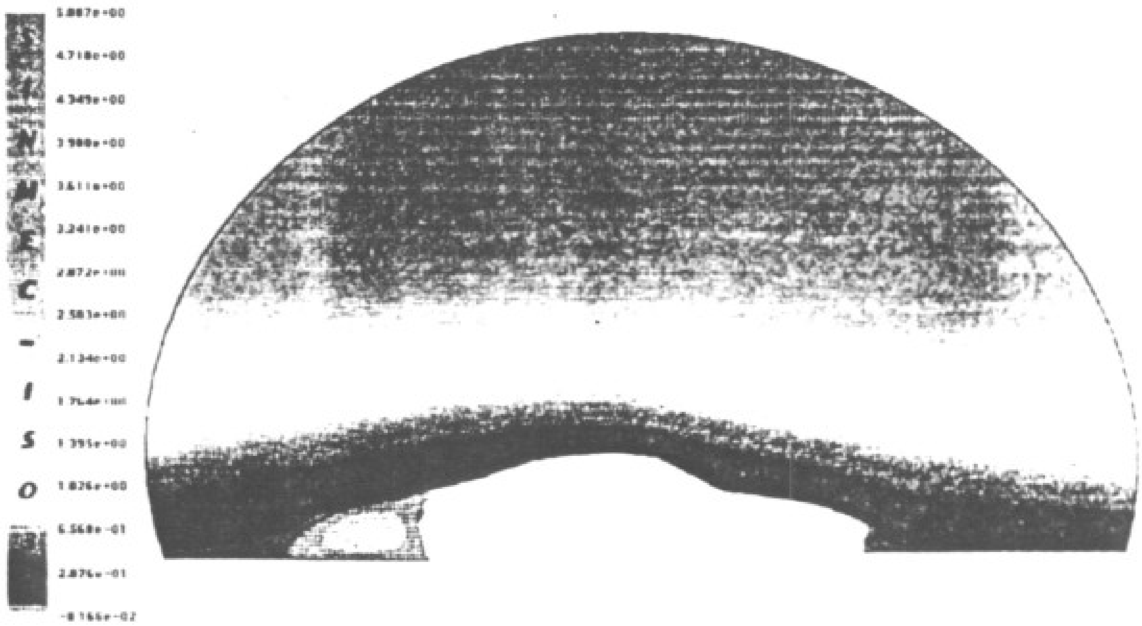
This brief report has shown that the CFD Laboratory is involved with several CPU time consuming activities related to CFD calculations. It is clear that the computer capabilities installed at the UFSC is far beyond the needs of our Laboratory. The expectation of using the CRAY at the Supercomputer Center at UFRGS did not became true due to the difficulty in assessing the facilities. We have received the best assistance and help from people at the Supercomputer Center, but it seems to us that the existing linkage is not fast enough to deal with massive transfer of data.

4. REFERENCES

1. Silva, A. F. C. and Maliska, C. R., " *Uma formulação Segregada em Volumes Finitos para Escoamentos Compressível ou Incompressível em Coordenadas Generalizadas*", **II ENCIT**, pp. 11-14, Brazil, (1988).
2. Maliska, C. R. and Silva, A. F. C., "*A Boundary-Fitted Finite Volume Method for the Solution of Compressible and/or Incompressible Fluid Flows Using Both Velocity and Density Correction*", **7th International Conference on Finite Element in Flow Problem**, pp. 405–412, Alabama, USA, (1989).
3. Marchi, C. H., Maliska, C. R. and Silva, A. F. C., "*A Boundary-Fitted Numerical Method for Solution of Three-Dimensional All Speed Flow Using Co-located Variables*", **III ENCIT**, pp. 351–356, Brazil, (1990).
4. Maliska, C. R., Cunha, A. R., Livramento, M. A. and Silva, A. F. C., "*Tridimensional Petroleum Reservoir Simulation using Generalized Curvilinear Grids*", **Submetido ao V ENCIT**, Brazil, (1994).
5. Cunha, A. R., Maliska, C. R., Silva, A. F. C. and Livramento, M. A., "*Two-Dimensional Two-Phase Petroleum Reservoir Simulation Using Boundary-Fitted Grids*",**Submetido ao V ENCIT**, Brazil, (1994).
6. Maliska Jr., C. R., "*Um Robusto Gerador de Diagramas de Voroni para Discretização de Domínios Irregulares*", **XIV Congresso Íbero Latino-Americano de Métodos Computacionais em Engenharia**, pp. 753-762, Brazil, (1993).
7. Marcondes, F., Zambaldi, M. and Maliska, C. R., "*Simulação Numérica de Reservatórios de Petróleo Utilizando Malhas de Voronoi*" **Submetido ao V ENCIT**, *Brazil, (1994)*.
8. *Maliska Jr., C. R. and Dihlmann, A.*, " *Visualizador Tridimensional para Campos Escalares e Vetoriais*" **XII CILAMCE"**, *pp. 354–363, Brazil, (1992)*.
9. *Maliska Jr., C. R. and Bezerra, L. H.*, "*Resolução Numérica de Problemas de Transferência de Calor em Diagramas de Voronoi*". **Aceito para publicação no XVII CNMAC**, *Brazil. (1994)*.

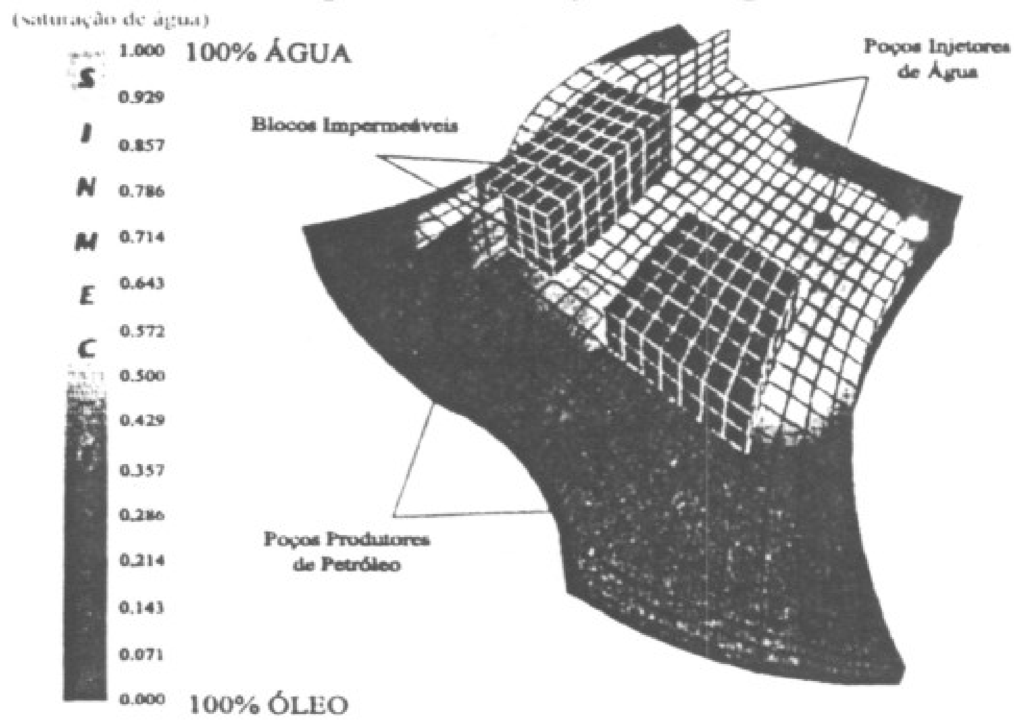
Simulação de Aerodinâmica

Escoamento Sobre um Automóvel
Movendo-se a 60 Km/h



Simulação de Reservatórios de Petróleo

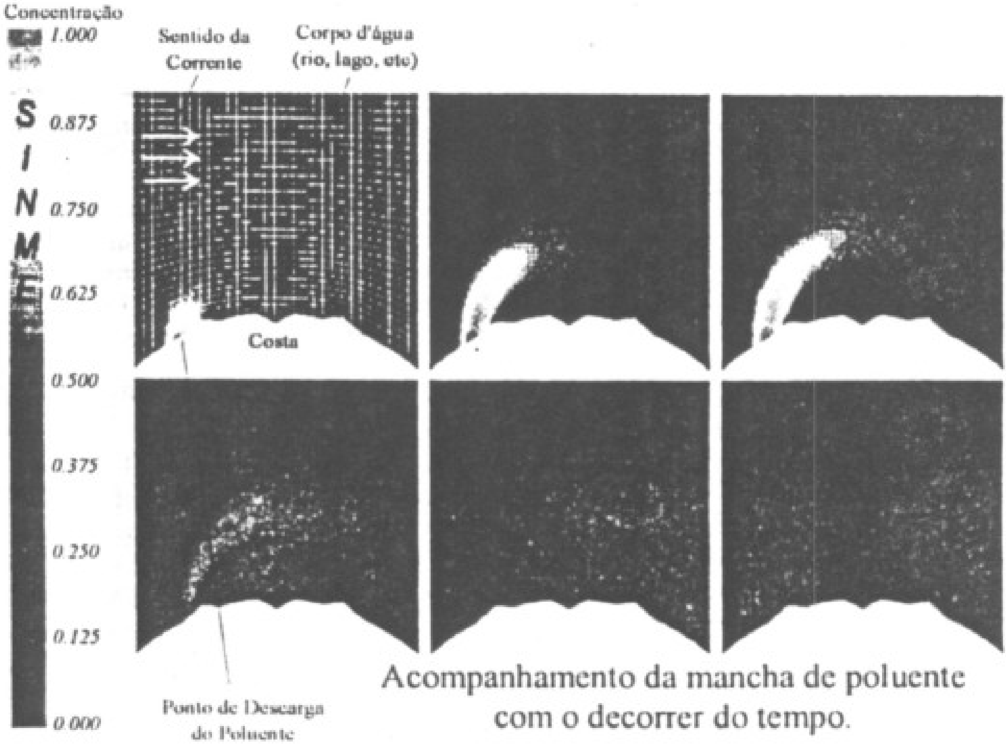
Campo de Saturação de Água



Figuras 1 e 2.

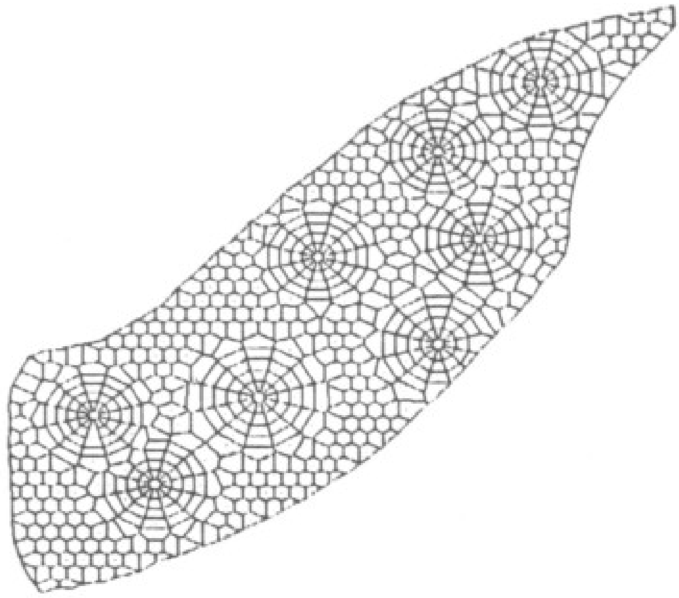
Simulação de Dispersão de Poluentes

Campo de Concentração de Poluente



Acompanhamento da mancha de poluente com o decorrer do tempo.

Diagrama de Voronoi para Simulação de Reservatórios de Petróleo



Figuras 3 e 4.