NON-NEWTONIAN FLUID FLOW ANALYSIS WITH FINITE DIFFERENCE AND FINITE VOLUME NUMERICAL MODELS

JURE MARN*, MARJAN DELIC, ZORAN ZUNIC

Faculty of Mechanical Engineering
University of Maribor
Smetanova 17, 2000 Maribor, Slovenia

*E- mail: jure.marn@uni-mb.si
Fax: x386.2.220.7990

Received: 19.4.2001, Final version: 26.10.2001

ABSTRACT:
Suitability of finite difference method and finite volume method for computation of incompressible non newtonian flow is analyzed. In addition, accuracy of numerical results depending of mesh size is assessed. Both methods are tested for driven cavity and compared to each other, to results from available literature and to results obtained using commercial code CFX 4.3.

ZUSAMMENFASSUNG:
Zwei numerische Methoden, die "finite difference" und die "finite volume" Methode zur Berechnung inkompressibler nicht-newtonischer Strömungen werden in diesem Beitrag diskutiert. Neben der grundsätzlichen Funktionsweise wird auch die Genauigkeit der numerischen Ergebnisse in Abhängigkeit von der Gittergrösse diskutiert. Am Beispiel einer Rohrströmung werden beiden Methoden unter Zuhilfenahme des Programm CFX 4.3 getestet und die erhaltenen Resultate mit der Literatur verglichen.

RESUMÉE:
La justesse de la méthode de différence finie et de la méthode de volume fini pour la modélisation de l'écoulement incompressible non Newtonien est analysée. De plus, la précision des résultats numériques, qui dépend de la taille du maillage, est mise en évidence. Les 2 méthodes sont testées pour le cas de la cavité conduite, et comparées. Elles sont également comparées aux résultats disponibles de la littérature et aux résultats obtenus en utilisant le programme commercial CFX 4.3.

KEY WORDS: Stream function-vorticity formulation, method characterization, driven caviy flow

1 INTRODUCTION

During past several decades the problem with flyash from fossil fuel power plants have become more and more important from both ecological and economical point of view. Largest of Slovenias coal mines, Velenje, has been extensively excavated for several decades. The coal in form of lignite leaves significant amounts of residue in form of ash, and flyash. This flyash is mixed with cement and chalk and used as input raw material for use in the main, for example, for reinforcing the floor of particular excavation layers in order to prevent premature slumping.

In order to numerically model flyash and water suspension pipeline transport one needs reliable non newtonian flow models and numerical models which are subject of presentation in this contribution. These models should be useful for those assessing pressure drop in one dimensional or lumped parameter computer code used for calculation of realistic flow data conditions using data from existing pipeline within coal mine. In this respect, large part of this paper is devoted to displaying differences between two widely used and recognized methods for treatment of partial differential equations: finite difference method (FDM), and finite volume method (FVM).

Description of both methods, their advantages and disadvantages can be found in texts on computational fluid dynamics, e.g. Garg, 1998 [3]. Therein, the authors describe the FDM as a method in which the partial derivatives appearing in the governing equations are replaced with algebraic difference quotients yielding a system of algebraic equations which can be solved for the flow-field variables at the specific, discrete grid points in the flow domain. On the other hand, FVM is a technique by which the integral formulation of the conservation laws are discretized directly in the physical space.

Figure 10: Comparison of profiles of horizontal velocity v_x through vertical centerline and vertical velocity v_y through horizontal centerline, FVM and FDM on 129 * 129 mesh and CFX on 129 * 129 and 257 * 257 mesh (Power Law using m = 0.0025, n = 1).

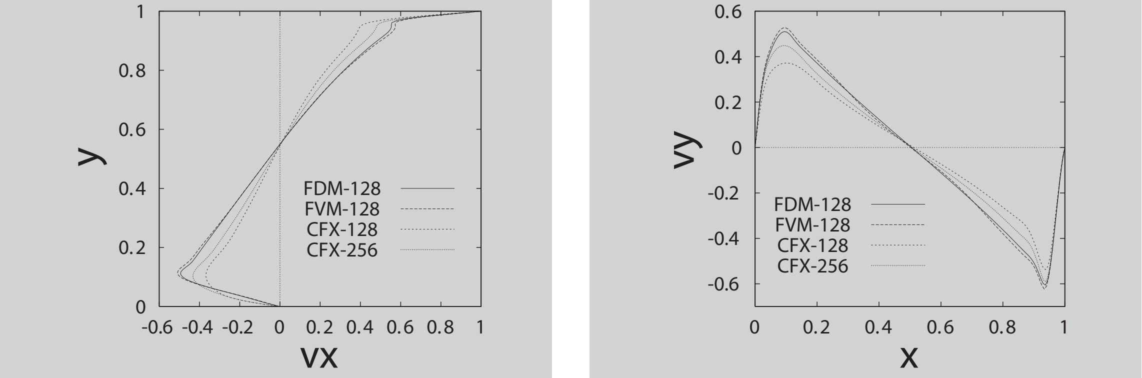
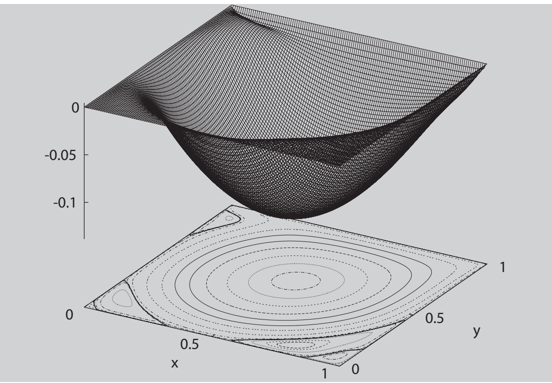


Figure 11: The flow field in driven cavity using 129 * 129 mesh size using finite difference method (Power Law m = 0.0001, n = 1.5).



computation. Parameter *n* has larger influence on Power Law as Carreau uses *n*/2 as evidenced by the velocity profiles.

3.3 COMPARISON OF THE MODELS TO COMMERCIAL CODE

The results were compared to commercial computer code *CFX 4.3* known for its robustness. For Power law, the results were compared for model parameters m = 0.0001 and n = 1.5 with mesh size of 129 * 129 nodes. Figure 10 shows that the velocity profiles of both models compare well one to another, and that the results obtained using commercial code (*CFX 4.3*) for mesh size 129 * 129 differ greatly from both models presented in this work. It should be noted that reducing the mesh size *CFX* results approach the results presented using *FDM* and *FVM*. *CFX* is based on *FVM* itself taking advantage of upwind method of the first order. Choosing method of higher order causes *CFX* solver to diverge as model parameters were quite demanding.

Using *CFX* to describe flows with higher viscosity or flows with less demanding model parameters is satisfactory. Hence, our initial goal of favorably comparing own codes to state of the art commercial codes for non newtonian flows was fulfilled. Figure 11 shows qualitative results for flow field for model parameters obtained using *FDM*. Figure shows six vortices. The main vortex covers most of the cavity, two pairs of vortices are formed in lower vertices of the cavity, and additional vortex in upper left corner of the cavity. One observes similar vortices in newtonian flows at higher *Re* numbers.

Based on the results presented one may conclude that both methods as presented in this work are suitable for real fluid analysis, in particular for fluids with high viscosity, e.g. slurries. The commercial program *CFX 4.3* is rather quick and robust yet one should perform nodalization analysis before actual use on particular problem. It should be noted that if convective terms govern the flow thus making the equations hyperbolic in nature one should refine the mesh thereby expose oneself to memory and CPU speed limitations.

4 CONCLUSION

This contribution deals with formulation method of finite difference (*FDM*) and finite volume methods (*FVM*) using method of characteristics for computation of viscous incompressible non newtonian fluid flow. The formulation have been checked using driven cavity. The effect of mesh size on accuracy of the results is shown as well as effect of model parameter *n* and comparison with available literature. The results of both methods as presented are also compared to commercial code program (*CFX 4.3*). Based on the results presented one may conclude that both methods describe quite well behavior in the cavity with slightly better results using *FVM*. The commercial code *CFX 4.3* gives less accurate results, however, it is user friendly and has good pre- and post- processor.

ACKNOWLEDGEMENT

The authors wish to acknowledge Ministry of Education, Science, and Sport of Republic of Slovenia for financial support, and Mr. Primoz Ternik and other coworkers of Laboratory for transport phenomena of Faculty of Mechanical Engineering at University of Maribor for help in using program *CFX 4.3*.

NOMENCLATURE

E - inviscid flux vector in x direction
E_{vis} - viscid flux vector in x direction
E - inviscid flux vector in ξ direction
E_{vis} - viscid flux vector in ξ direction

This is an extract of the complete reprint-pdf, available at the Applied Rheology website
http://www.appliedrheology.org

G - inviscid flux vector in y direction
G_{vis} - viscid flux vector in y direction
G - inviscid flux vector in η direction
G_{vis} - viscid flux vector in η direction
J - inverse Jacobian determinant
L - characteristic length
m, n - model parameters
p - pressure
Q - vector of primitive variables in Cartesian coordinate system
Q - vector of primitive variables in curvilinear coordinate system
t - time
v_i, v_x, v_y - velocity components
V - characteristic velocity
x, y - Cartesian coordinates
β - artificial compressibility
$\dot{\gamma}$ - shear rate
η - dynamic viscosity
η_0 - zero shear rate dynamic viscosity
η_∞ - infinity shear rate dynamic viscosity
λ - wave speed
λ_j - Riemann matrix eigenvalue
ν - kinematic viscosity
ρ - density
τ - shear stress
χ - time constant
ψ - stream function
ω - vorticity

REFERENCES

[1] B. C. Bell, K. S. Surana: p-Version Least Squares Finite Element Formulation for Two-dimensional, Incompressible, Non-newtonian Isothermal and Non-isothermal Fluid Flow, International journal for numerical methods in fluids 18 (1994) 127-162.

[2] D. Drikakis, P. A. Govatsos, D. E. Papantonis: A Characteristic-Based Method for Incompressible flows, International journal for numerical methods in fluids 19 (1994) 667-685.

[3] V. K. Garg (editor): Applied Computations Fluid Dynamics, Marcel Dekker, Inc. 1998.

[4] U. Ghia, K. N. Ghia, C. T. Shin: High-Re Solutions for Incompressible Flow Using the Navier-Stokes Equations and a Multigrid Method, Journal of computational physics 48 (1982) 387-411.

[5] C. Hirsch: Numerical Computation of Internal and External Flows, John Wiley & Sons, 1988.

[6] J. Marn, M. Delic: Analiza toka newtonskih in nenewtonskih tekocin s tokovno funkcijo in vrtincnostjo, Strojniski vestnik 45/2 (1999) 47-56.

[7] J. Marn, L. Skerget, M. Delic: Non Newtonian driven cavity flow comparison between boundary element and finite difference methods, Second International Conference on Advances in Fluid Mechanics, Udine, Italy, (1998) 261-270.

[8] Schetz, J.A., Fuhs, A.E.: Handbook of fluid dynamics and fluid machinery, Wiley Interscience, USA, 1996.

[9] Z. Zunic: Numericna obravnava problema aerodinamike vozil, Magistrsko delo, Univerza v Mariboru, Fakulteta za strojnistvo, 1997.

This is an extract of the complete reprint-pdf, available at the Applied Rheology website
http://www.appliedrheology.org