18th International Conference
ENGINEERING MECHANICS 2012
Svratka, Czech Republic, May 14 – 17, 2012

pp. 30–31
Paper #345

IMPLEMENTATION OF DIRECT NUMERICAL SIMULATION OF VISCOUS INCOMPRESSIBLE FLOW

M. Beňo*, B. Patzák**

Abstract: *The method for solving viscous incompressible flows, based on Glowinski, Pan,Hesla & Joseph (1998), is presented for the direct numerical simulation of viscous incompressible flow. It uses a finite element discretization in space and an operator-splitting technique for discretization in time. Quadratic approximation is employed for velocity flow and linear approximation for pressure on triangle elements. The goal is to develop more efficient and accurate numerical tool for computing viscous flows. The accuracy of the presented method has been confirmed on two common cases by implementation in Matlab program: the Poiseuille flow test, and on driven cavity flow test.*

Keywords: ***finite element, liquid flow, operator splitting, Navier-Stokes equations.***

1. Introduction

The aim of this article is to present a method for solving viscous incompressible flows. The finite element implementation is based on the incompressible Navier-Stokes equations on structured two dimensional triangular meshes with using operator splitting for time discretization (Glowinski 2003). The fractional-time-step scheme described by Marchuk (1990) has been employed. Liquid is supposed to be incompressible and Newtonian. The advantage of this method is that no repeated remeshing is required. Also the assembled mass matrix remains constant and so it does not have to be assembled at every time step. By splitting one time step into three successive substeps the discretization enables better approximation of results. Pressure is computed in this case in the first sub-step while velocity is computed at each substep. To discretize the domain the Taylor-Hood triangular elements have bed used with second degree polynomic approximation of velocity and first degree polynomic approximation of pressure.

2. Numerical validations:

The matlab implementation of the presented method has been verified using two benchmark tests: the flow between parallel plates (Poiseuille test) where the quadratic behavior of the velocity has been verified, and on driven cavity flow test.

2.1. Poiseuille test in tube:

In this classic test, the steady state velocity and pressure distributions is simulated for a fluid moving laterally between two plates whose length and width is much greater than the distance separating them. The geometry of the studied domain has been divided into 400 elements. The perfect friction on both plates (zero velocity) has been assumed. The results agree with the analytic solution.

2.2. Driven Cavity flow

The flow inside closed cavity has been simulated and compared to the results presented in literature (Botella and Peyret, 1998 ; Rabenold, 2005). The viscosity was set to $\eta = 10^{-2} Pa\,s$, and the Reynold's number is computed as $1/\eta$ (based on geometry of size 1 and maximum velocity 1). The

* Ing. Matej Beňo: Faculty of Civil Engineering, Czech Technical University; Thákurova 7; 166 29, Prague; CZ, e-mail: matej.beno@fsv.cvut.cz

** Prof. Dr. Ing. Bořek Patzák: Faculty of Civil Engineering, Czech Technical University; Thákurova 7; 166 29, Prague; CZ, e-mail:borek.patzak@fsv.cvut.cz

following parameters has been used: mass density $\rho = 1.0\ kg/m^3$, viscosity $\eta = 10^{-2} Pa\ s$, and the time step $\Delta t = 0.01$. The results are in good agreement, even though the present values are restricted to 30x30 grid points (see table No. 1).

Tab. 1: Velocity extreme through cavity centerlines at $Re = 100$

method	u_{min}	v_{min}	v_{max}
Present	-0.2162	-0.2493	0.1771
Botella and Peyret	-0.21279	-0.25266	0.17854
Rabenold	-0.2140424	-0.2538030	0.1795728

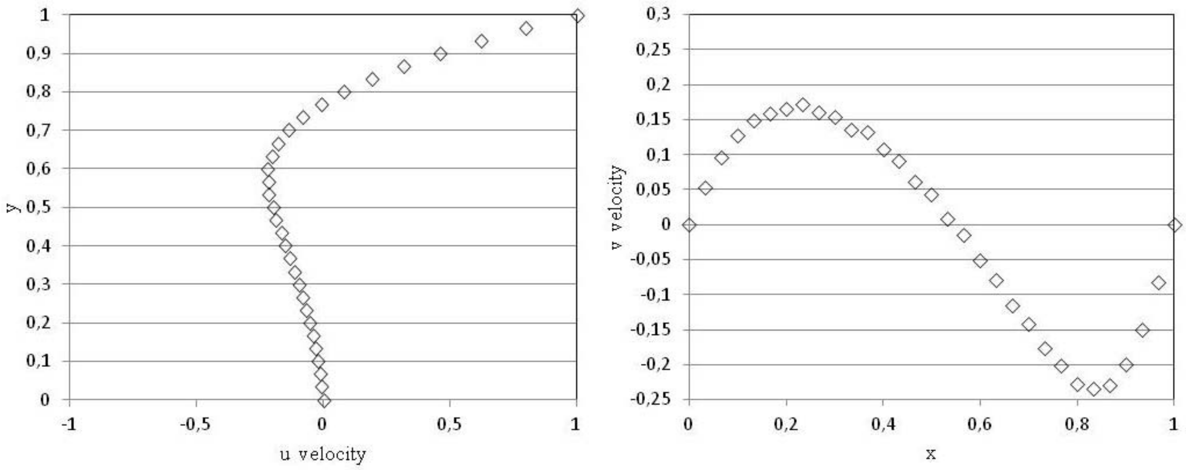

Figure 4: Velocity profiles through cavity centerlines at $\eta = 10^{-2} Pa\ s$ and 30 x 30 grid size.

Conclusion

Presented work describes formulation and implementation of non-stationary, incompressible flow finite element solver. The Tayler-Hood elements (P2-P1) have been implemented. For time discretization the operator splitting method is used and it reduces computation of velocity and pressure together to one time substep whereas in other substeps only velocity is being solved. The model is verified using standard benchmark tests from literature.

References

Botella, O., Peyret, R. (1998) Benchmark spectral results on lid-driven cavity flow, *Computers&Fluids,* 27,4, pp. 421-433

Glowinski, R., Pan, T.-W., Hesla, T.I., Joseph, D.D. (1998) A distributed Lagrange multiplier-fictitious domain method for particulate flows. *Int. J. Multiphase Flows* 25, pp. 755–794.

Glowinski, R. (2003) Finite Element Methods for Incompressible Viscous Flow, in: *Handbook of Numerical Analysis*, vol. 9 (P.G. Ciarlet, J.L. Lions eds), North-Holland, Amsterdam, pp. 3-76.

Marchuk, G.I (1990) Splitting and Alternating Direction Methods, in *Handbook of Numerical Analysis,* vol. 1 (P.G. Ciarlet, J.L. Lions eds), North-Holland, Amsterdam, pp. 197-462.

Rabenold, P.T (2010) *Lid-driven flow in a rectanglar cavity with corner obstacles*, available on-line http://www-users.math.umd.edu/~rabenold/enme646_project.pdf