SindhUniv. Res. Jour. (Sci. Ser.) Vol. 48 (2) 419-422 (2016)

SINDH UNIVERSITY RESEARCH JOURNAL(SCIENCE SERIES)

Scientific Computation of Rotating Annular Flow of Newtonian Fluid

S. BHATTI[++], M. E. BALOCH*, I. Q. MEMON,

Department of Basic Sciences and Related Studies, Mehran UniversityEngineering and Technology, Jamshoro

Received 10th July 2015 and Revised 09thFebruary 2016

Abstract: A finite element numerical schemes presented in pseudo three–dimensions to simulate the isothermal flow of incompressible Newtonian fluid. The problem addressed in this research work is an annular pipe with fixed outer pipe and rotating inner cylinder, which has numerous industrial importance. For primitive variables a full Navier–Stokes equation computed in a cylindrical polar co–ordinate system. Whereas, domain is discredited in two–dimensions. The time derivative is discredited through multi step Taylor series expansion. Numerical scheme adopted is Taylor–Galerk in with combination of second order pressure–correction method in its semi–implicit form. The main focus of this article is to analyse rotating flow through annular configurator domain. Flow structure is presented through velocity vector field and pressure distribution to investigate impact of rotation of inner cylinder and effects of increasing inertia. Predicted numerical results on pseudo three–dimensional algorithm, found in good agreement with other numerical and experimental solutions.

Keywords: Annular Flow, Newtonian fluid, Finite Element Method, Pseudo Three Dimensions, Taylor–Galerkin/pressure–Correction Algorithm.

1. INTRODUCTION

For optimal designing of equipment‘s in processing industries, numerous material processes are involved. Many industries, such as, ceramics, cosmetic, composite material, food, plastic, polymer, refinery, textile, and other chemical industries presents many challenging problems (Champion, 1992). In all above processing industries macroscopic behaviour of material is dominant effect on the production, therefore, proper inside knowledge of dynamics is very important to investigate. Also material properties, complexity of geometry and optimal power consumption give impetus to researchers to study the annular flow (Chhapra, 1992).

An experimental and numerically study has been presented on the heavy oil transportation in lubricating water (Joseph, *et al* 1997). Study is recognised the capacity of energy efficiency, rising models, industrial experience, stability and give future directions of investigation (Joseph, 2009).

A close form for annular flow has been realised (Pinho, 2000). Study is focused on viscoelastic fluid adopting a non–linear rheological PTT constitutive model (Thien, 1977) as this model represents both polymer melts and solution. Utilising a fully developed boundary conditions, through radial cavity, the variation in stresses and kinematic of the fluid has been investigated. For given flow rate, the exact solutions vindicate the pressure drop scaled is free from ratio between inner and outer diameters.

For fully developed, an incompressible laminar flow of constant viscosity fluid past an eccentric annulus along with rotating inner cylinder and collective bulk axial flow, a numerical and an experimental investigation has been presented (Escudier, *et al.,* 2000). For various eccentricity values and Taylor number, the results demonstrate the flow structure, friction factor and wall shear–stress distribution. It has been shown that the radial/tangential velocity field has direct impact on the axial velocity, due to rotational velocity of inner cylinder therefore, there is a strong influence on the flow structure. This rotational influence of inner rotational cylinder on the axial flow velocity is reflected on the behaviour of wall shear–stresses, at both inner cylinder and outer pipe, and thus on friction factor as well. Unpredictably, at fixed Reynolds number and large values of eccentricity ($\varepsilon > 0.9$)enhancing Taylor number the friction factor rising instead of displaying diminishing tendency.

Pressure driven axial incompressible flow in concentric cylinders with rotating central cylinder, the surgical system of thread injection of medical grafts has been modelled (Walton, 2003). At asymptotically large Reynolds number, the nonlinear nature of basic flow is analysed theoretically. It has witnessed that non–axisymmetric finite–amplitude neutral modes can be

[++]Corresponding author: email: S. BHATTI
*Institute of Computer Engineering, University of Duisburg–Essen, Campus Duisburg, Germany.mahera_erum2007@yahoo.co.uk

supported over a wide range of thread radii and injection velocities. The axial force on the thread has been computed, identified that is considerably fewer than anticipated in undisturbed–flow theory. Computed solutions confirm with thread–annular experimental observations.

In a turbulent flow study through concentric annulus along with moving inner cylinder at three different rotational velocities, (Chung, 2005).The turbulent flow structure close to the inner wall has been analysed using large–eddy model. The central theme of the research work is to investigate the weakening of turbulent flow structure near–wall,due to rotation of the inner cylinder.

Spinning annulus flow has been investigated in the gap between concentric circular cylinders (Townsend, 1987). Spinning was developed in the flow with rotation of the inner cylinder and an axial pressure gradient acting at the same time. Imposing more axial force on the cylinder walls resulted in a restructuring of velocity profile in axial direction, without losing fundamental qualitative characteristic of the flow structure. Investigation has been limited to flows generated through rotation of the outer tube, and axial motion of the inner cylinder alone. It has been perceived that both conditions stimulated to stabilise the flow. The structure and position of the crucial layer that appeared in this flow when it is undergoing transition was the focus of interest of study.

2. PROBLEM SPECIFICATION

The problem investigated in this article is the flow fluid in an axisymmetric annulus. For numerical techniques, simple annular flow is a benchmark problem regarding the accuracy, stability and convergence. However, rotating annular flow presents many interesting features particularly at high Reynolds number. Flow domain with boundary condition is displayed in **(Fig-1)**. Solid inner cylinder is rotating along azimuthal direction with no–slip in axial and radial directions while, fluid is entering in the annulus at inlet with no cross flow, i.e. between outer pipe and inner rotating cylinder. On outer pipe wall no–slip boundary conditions are enforced. While, zero initial conditions are imposed on all primitive variables. At exit, vanishing pressure datum and velocity components. Steady–state results are achieve adopting time–dependent finite–element algorithm.

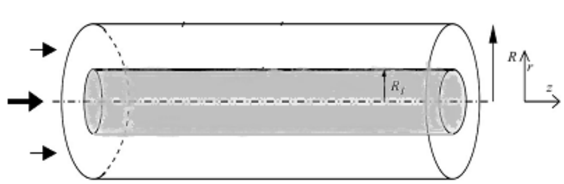


Fig.1:Computational domain of annular flow.

The length of the flow geometry of the annulusis five (05) non–dimensional units, while, the radius of inner rotating cylinder is taken half (0.5) and the radius of outer pipe is taken one (01). Whereas, computational domain isdiscretiseinto ten (10) elements in radial direction and one hundred (100) in axial direction. While, The total number of elements are one thousands (1000) andtotal number of vertex nodes are one thousand one hundred eleven (1111).

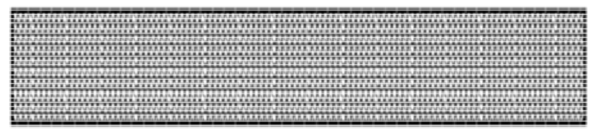


Fig. 2:Finite element mesh of annular flow.

3. GOVERNING EQUATIONS AND NUMERICAL SCHEME

Consider the incompressible flow of constant viscosity Newtonian fluid through axisymmetric annulus under isothermal condition. Inner cylinder is rotating at constant velocity, whereas, fluid is moving in annulus tube axially. For pseudo three–dimensional mathematical formulation, annulus flow governs by the conservation of mass and momentum transport equations. System of equations comprises velocity–pressure primitive variables. Flow is considered in axisymmetric cylindrical polar coordinate's frame of reference. Present study employ pseudo three–dimensions, the components of velocity vector field are described as $\boldsymbol{v}=\left(v_r,v_\theta,v_z\right)$, where, '$v_r$' is radial velocity, '$v_\theta$' is azimuthally velocity and 'v_z' is axial velocity while the spatial vector is considered as $\vec{x}=\left(r,0,z\right)$. The dimensionless without body force as

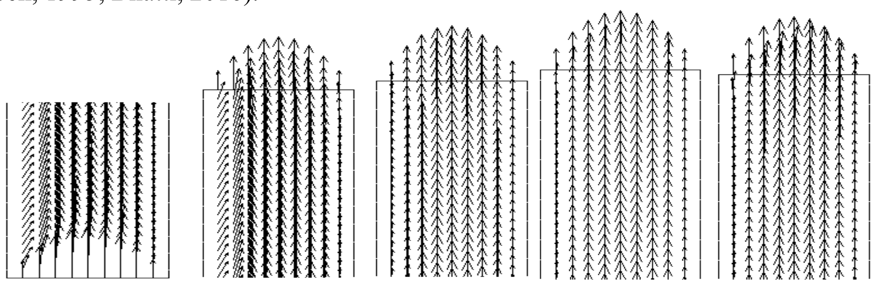
$$\nabla.\boldsymbol{v}=0, \qquad (1)$$

$$\frac{\partial \boldsymbol{v}}{\partial t}+(\boldsymbol{v}.\nabla)\boldsymbol{v}=\frac{1}{\mathrm{Re}}\nabla^2\boldsymbol{v}-\nabla p, \qquad (2)$$

A time–marching finite–element numerical weighted residual technique is adopted in this research work. Temporal domain is discretised through Taylor series expansion (Townsend, 1987). To obtain second order accuracy in time, series is truncated up to third term, whilst, spatial domain is discretised adopting Galerkin approximation in combination of pressure correction method. Employing Lax–Wendroff approach (Donea, 1984), time difference is divided into two steps. First step estimate esthenumerical result sat half time–step from initial condition employing forward difference in time. While, on step two central differences is adopted and this step is further distributed in three stages using Crank–Nicolson on pressure term. At stage one, intermediate non–solenoidal velocity vector field is computed, at stage–2, pressure–difference is calculated and at final stage–3 for full time–step divergence–free velocity field computed. Implicitness is applied on only diffusion term to adopt higher time step and accelerate the convergence of the procedure. At stage–1 and stage–3 are governed by the mass matrix and solved by modified Jacobi's method, while,stage–2 is governed by the pressure difference stiffness matrix which is computed by direct Choleski's method. For further details on numerical scheme readers are referred (Baloch, 1994, Baloch, 1995, Bhatti, 2016).

4. NUMERICAL RESULTS

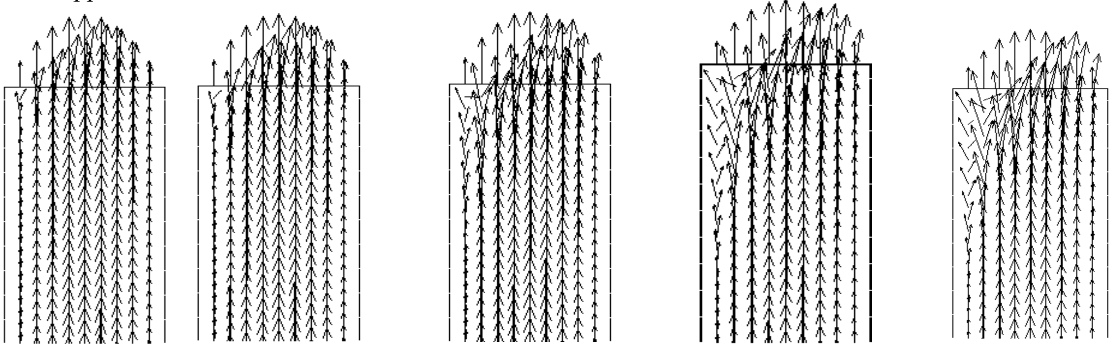
In **(Fig-3a)**, the inlet parabolic axial velocity profile is illustrated. At various Reynolds numbers (Re = 01, 10, 50, 100, 250, 500, 570, 800), the axial velocity profile vindicates that the fluid is revolving along the wall of rotating inner cylinder. In **(Fig.3, Fig.4)**, at various inertial values (Re = 01, 10, 50, 100, 250, 500, 570, 800), the outlet velocity distribution of the annulus is displayed.



(a) (b) (c) (d) (e)

Fig. 3:Velocity vectors of annulus flow: (a) lower inlet part of annuluswith parabolic boundary conditions,rotating right side inner cylinder and fixed outer pipe wall, (b) Re = 01, (c) Re = 10, (d) Re = 50 and (e)Re = 100.

The flow of fluid is spinning beside rotating inner cylinder wall and it behaviour is laminar at the Reynolds number 01,10and 50.its observed in this study that when the Reynolds number increase (at 100, 200, 250) the rotation of the flow at the outlet side of the inner rotating cylinder wall will increased and the behaviour of the flow slightly changed. At the Reynolds number 500, 750 and 800 the behaviour of flow is changed from laminar to turbulence at the along the right corner (or the wall of an inner rotating cylinder) at the outlet and the maximum rotation is appeared here.



(a) (b) (c) (d) (e)

Fig.4: Velocity vectors of annulus flow with increasing inertia:(a) Re = 200, (b) Re = 250, (c) Re = 500, (d) Re = 570 and Re = 800.

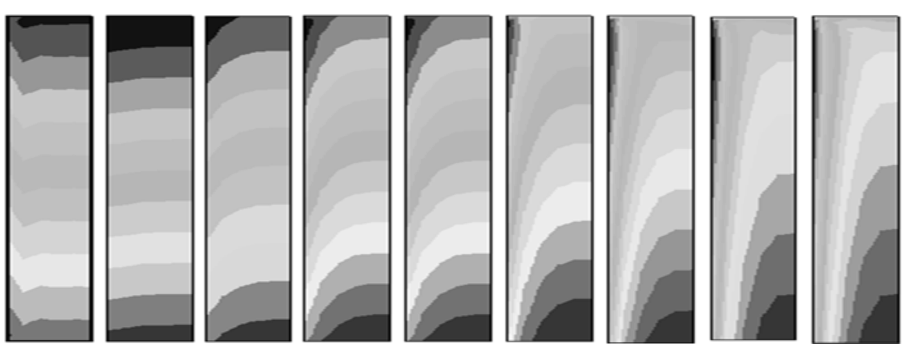


Fig.5: Pressure isobars of annulus flow with vanishing pressure fixed at exit upper part,rotating right side inner cylinder and fixed outer pipe wall:

For rotating annulus flow, in **Fig. 5(a)–(i),** pressure contours are presented with increasing inertia from left to right (Re = 10 to 800). Behaviour of the pressure clearly demonstrates that at low Reynolds number, pressure increase linearly except in the vicinity of rotating cylinder. While, at high Reynolds number, due to rotation of inner cylinder non–linear behaviour of pressure is observed.

5. CONCLUSION

It is concluded that developed pseudo three–dimensional algorithm in cylindrical polar coordinates perform very well and numerical solutions are in good agreement with other numerical and experimental results. From predicted solutions is observed that at low values of inertia, i.e., Reynolds number of order one, incompressible Newtonian fluids behave smooth and laminar. The impact of rotational motion of inner cylinder is observed very low. Whilst, at high Reynolds number of order two, flow become translate and rotate with increasing Reynolds number further, (say Re=100–250). While, inertia increase beyond Re = 250, flow somewhat change and become turbulent. For high Reynolds numbers, (Re = 500–800), at the outlet of the pipe the flow completely rotate in the annulus and turbulence is developed. Whereas, the pressure in the annulus is increasing with increasing Reynolds number.

6. ACKNOWLEDGEMENT

First author greatly acknowledges Late Prof. Dr. Rana Khalid Naeem for his supervision and encouragement, also the department of Mathematics for providing computing facilities, University of Karachi, Sindh.

REFERENCES:

Baloch, A., (1994) “Numerical Simulation of Complex Flows of Non–Newtonian Fluids”, Ph. D. Thesis, University of Wales Swansea.

Baloch, A. and M. F. Webster, (1995) ‘A computer simulation of complex flows of fibre Suspensions, Computer Fluids 24 (2), 135-151.

Baloch, A., P. Townsend, and M. F. Webster, (1996) ‘On the Highly Elastic Non–Newtonian flows’, J. Non–Newtonian Fluid Mech. 75, 139-166.

Bhatti, S. D., (2016) ‘Numerical Simulation of Rotating Annular Flow of Newtonian Fluid’, M. Phil. Thesis, Department of Mathematics, University of Karachi.

Champion, E. R., (1992) “Finite Element Analysis in Manufacturing Engineering”, McGraw Hill. Company Inc. USA.

Chhapra, R. P., (1992) “Bubbles, Drops, and Particles in Non-Newtonian Fluids”, CRP press London.

Chung, S. Y., H. J. Sung, (2005) ‘Large-eddy simulation of turbulent flow in a concentric annulus with rotation of an inner cylinder’, International journal of heat and fluid flow, 26(2), 191-203.

Donea, J., (1984) “A Taylor-Galerkin Method for Convective Transport Problems”, Int. J. Num: Methods. 20,101Pp.

Escudier, M. P., I. W. Gouldson, P. J. Oliveira and F. T. Pinho, (2000), ‘Effects of inner cylinder rotation on laminar flow of a Newtonian fluid through an eccentric annulus’, International J. of Heat and Fluid Flow, 21(1), 92-103.

Hussong, J., N. Bleier and V. I. V. Ram, (2006) ‘The structure of the critical layer of a swirling annular flow in transition’, Institute for Thermo- und Fluiddynamik, Ruhr Universität Bochum, 44780 Bochum, Germany.

Joseph, D. D., R. Bai, K. P. Che and Y. Y. Renardy, (1997),‘Core-annular flows’, Annual review of fluid mechanics, 29(1), 65-90.

Joseph, M. R., (2009) “Numerical Simulation of Two–Phase Annular Flow”, Ph.D. Thesis, Rensselaer Polytechnic Institute Troy, New York.

Pinho, F. T. and P. J. Oliveira, (2000) Axial annular flow of a nonlinear viscoelastic fluidan analytical solution’, J. of Non-Newtonian Fluid Mechanics, 93(2), 325-337.

Thien, N. P. And R. I. Tanner, (1977) A new constitutive equation derived from network theory’, J. of Non-Newtonian Fluid Mechanics. 2(4): 353-365.

Townsend, P., M. F. Webster, (1987) ‘An Algorithm for three dimensional flows’, Paper T 12/1 NUMETA 87 Swansea U. K. Eds. Pande G. N. and Middleton, J. Nijhoff. Dordrecht Holland.

Walton, A. G., (2003)‘The nonlinear instability of thread–annular flow at high Reynolds number’, J. of Fluid Mechanics, Vol. 2.45-52.