

# Simulations of Three-dimensional Second Grade Fluid Flow Analysis in Converging-Diverging Nozzle

Ahmed S\* and Bano Z

Department of Mathematics, Sukkur Institute of Business Administration, Sukkur, 62500, Pakistan

## Abstract

Very little work have been reported on the computational methods for non-Newtonian fluid turbulent flows. This is due to nonlinear system of elliptic partial differential equations that makes the solution very difficult. Another concern is the meshing of flow domain that accounts for complications in solving most problems. In this work we present standard Galerkin finite element method for the steady incompressible non Newtonian fluid flow in a converging-diverging nozzle. The flow is fully three-dimensional with turbulent characteristics. The main aim is to study the velocity and shear stress profiles. Shock profiles are noted for specific pressure boundary conditions. Moreover the plotted results shows variations of velocity components, pressure and turbulence energy dissipation. It is observed that for non-Newtonian fluid flow the mass flow rate and pressure loss effects are significant in diverging part of nozzle.

**Keywords:** Computational; Newtonian fluid flows; Unsteady; Three-dimensional

## Introduction

Analyzing flow patterns in a converging-diverging nozzle has been one of interesting topic in computational fluid dynamics. There are numerous applications of this flow phenomena in aerospace and engineering sciences. Such process are difficult to handle analytically due to complex mathematical model associated to the flow and ensuing instabilities carried by flow parameters. Looking back to the history Jaffery [1] and Hamel [2], in their studies considered the converging diverging channel steady two dimensional Newtonian fluid flow. They observed quiet interesting results by treating Navier-Stokes equations with similarity transforms. Further developments were presented in Schlichting [3] and Batchelor [4] based on the boundary layer approximations. Makinde [5] examined the incompressible Newtonian fluid flow by incorporation of linearly diverging symmetrical channel. Recently, Zarqa et al. [6] performed approximate analytical analysis using Adomian decomposition method for a channel with variable diverging ratio. It is evident by several studies that the mechanism of such flows is characterized due the fact that shocks instabilities are produced within the flow domain. Mehta et al. [7] formulated finite element method for solid rocket motor. He worked on the transient flow with axisymmetric and anisotropic model on throat nozzle. Later on Nikhi et al. [8] presented the computational analysis for flow in da Laval nozzles. Apart from the flow analysis, there is significant contribution by researchers on the shape optimization analysis for nozzle flows [9].

Although many fluids in real world applications carry Newtonian behavior, there is a number of fluids that exhibit non-Newtonian behavior. The applications ranges from industrial materials like clay coatings, doilling muds, suspensions, oils, greases, polymer melts to biological materials. Because of the great diversity in the physical structure of non-Newtonian fluids there are many constitutive models that represent different types of non-Newtonian fluids. Among these, the second grade fluids exhibits a linear relation between the stress and the Rivlin-Ericksen tensor, its square and the second Rivlin-Ericksen tensor [10]. The resulting mathematical model is more complex than that for Newtonian fluid. To respond to this concern we opt for computational techniques to investigate the flow

parameters. The theoretical foundations of finite element method for non-Newtonian fluid flow is presented by Adelia [11]. They presented finite element method for elliptic type system of partial differential equations that represents steady state incompressible second grade fluid flow. Numerical testing of the proposed method has not reported yet. It is due to computational complexity of the mathematical model. A large number of solution techniques have been developed to solve such non-linear problems, these include approximate analytical techniques such as perturbation analysis, Homotopy analysis, adomian decomposition and numerical techniques such as finite difference method, finite volume method, finite element methods, operator splitting methods and spectral methods. The finite difference method is strongly influenced by meshing quality problem while finite volume method requires additional treatment for analyzing different flow regimes. The flexibility of meshing and segregated formulation of pressure and velocity in terms of the basic functions are the most desirable features of finite element method that makes it suitable and superior for investigating non-Newtonian fluid flow in converging-diverging channel. Another concern is the convergence and stability of numerical methods. Finite element method is not just suitable in terms of efficiency but also there is a sound stability analysis before implementation. This work addresses the application of finite element method for second grade fluid flow in a converging-diverging channel. We present two different results for two different set of primitive variables in order to see changes in flow regimes.

The remaining article is organized as follows. Section 2 presents basic equations for the flow problem and corresponding weak formulation. The numerical methodology and simulation set up

**\*Corresponding author:** Ahmed S, Department of Mathematics, Sukkur Institute of Business Administration, Sukkur, 62500, Pakistan, Tel: 071-5630225; E-mail: [dr.sidrah@iba-suk.edu.pk](mailto:dr.sidrah@iba-suk.edu.pk)

**Received** August 02, 2016; **Accepted** August 24, 2016; **Published** August 31, 2016

**Citation:** Ahmed S, Bano Z (2016) Simulations of Three-dimensional Second Grade Fluid Flow Analysis in Converging-Diverging Nozzle. J Appl Computat Math 5: 320. doi: 10.4172/2168-9679.1000320

**Copyright:** © 2016 Ahmed S, et al. This is an open-access article distributed under the terms of the Creative Commons Attribution License, which permits unrestricted use, distribution, and reproduction in any medium, provided the original author and source are credited.

are included in section 3 along with results and discussions. Finally concluding remarks are summarized in section 4.

## Basic Equations

The mathematical model represents the three dimensional flow of incompressible second grade fluid. It is to be noted here that the fluid material is incompressible, i.e. density is kept constant. The body forces and thermal effects are ignored.

Hence the dynamics are represented as:

$$\text{div } V = 0 \quad (1)$$

$$\rho \dot{V} = \text{div } T, \quad (2)$$

where  $\rho$  is density,  $V$  stands for three dimensional velocity vector and  $T$  is the stress tensor defines as:

$$T = -pI + \mu A_1 + \alpha_2 A_2 + \alpha_1 A_1^2 \quad (3)$$

With  $A_1$  and  $A_2$  be the first and second Rivlin-Ericksen tensors,  $\mu$  is the coefficient of viscosity and  $\alpha_1$  and  $\alpha_2$  are the normal stress moduli. Thus the equations that controls the transport of momentum with the fluid flow reads as:

$$0 = -\nabla p + \mu \Delta^2 V + (\alpha_1 + \alpha_2) \text{div} A_1^2 + \alpha_1 \left[ \Delta^2 V + \Delta^2 (\nabla \times V) \times V + \left| \nabla \left( V \bullet \nabla^2 + \frac{1}{4} A_1^2 \right) \right| \right] \quad (4)$$

The coupled equations 1 and 4 constitute a nonlinear elliptic system of PDEs. Moreover, boundary conditions are associated in order to have a well-posed problem. A rigorous weak formulation is given next. The flow domain  $\Omega$  is three-dimensional and  $\partial\Omega$  represents boundary of the domain. In this case it comprises of an inlet, outlet and curved surface of nozzle. The domain is discretized into non-overlapping cells. In order to derive weak formulation for the system (1-3) together with appropriate boundary condition, multiply the equations by a test function and then integrate over a finite element cell. The test function is taken in a Sobolove space with the property that it vanishes on the prescribed boundary. The solution is sought in terms of trial functions which satisfies the prescribed boundary conditions. For the continuity equation we get:

$$\int_{\Omega} q \cdot \text{div} V d\Omega = 0 \quad (5)$$

Defining  $\vec{u} = (u_1, u_2, u_3)^T$  with  $u_1$ ,  $u_2$  and  $u_3$  be the separate test functions for momentum equations, it follows:

$$0 = \int_{\Omega} \left[ -\nabla p + \mu \Delta^2 V + (\alpha_1 + \alpha_2) \text{div} A_1^2 \right] \cdot \vec{u} d\Omega \quad (6)$$

Consider the second integral on right hand side:

$$\int_{\Omega} (\mu \Delta^2 V) \cdot \vec{u} d\Omega = \int_{\Omega} \sigma \cdot \nabla \vec{u} d\Omega - \int_{\Gamma} (u_n \sigma^{nn} + u_t \sigma_{tt}) d\Gamma \quad (7)$$

Where  $\Gamma\Omega$  denotes the boundary of  $\Omega$ ,  $u_n$  and  $u_t$  are the normal and tangential components of  $\vec{u}$  respectively. Next is to define the basic functions  $\Psi_i(x)$  and  $\phi_j(x)$  for pressure and velocity components respectively. The approximation for  $V$  and  $p$  will be:

$$p_h = \sum_{i=1}^m p_j \Psi_j(x) \quad (8)$$

and

$$\vec{V}_h = \sum_{j=1}^n v_{1j} \phi_{j1}(x) + v_{2j} \phi_{j2}(x) + v_{3j} \phi_{j3}(x) = \sum_{j=1}^{3n} v_j \phi_j(x) \quad (9)$$

with

$$\phi_{j1} = \begin{pmatrix} \phi_j(x) \\ 0 \\ 0 \end{pmatrix}, \phi_{j2} = \begin{pmatrix} 0 \\ \phi_j(x) \\ 0 \end{pmatrix} \text{ and } \phi_{j3} = \begin{pmatrix} 0 \\ 0 \\ \phi_j(x) \end{pmatrix} \quad (10)$$

Working according to standard Galerkin FEM approach, we obtain:

$$\int_{\Omega} \Psi_j \text{div} \vec{V}_h d\Omega = 0, j = 1 : m, \quad (11)$$

$$\int_{\Omega} [\mu A_1 + \alpha_1 A_1 + \alpha_2 A_2] \cdot \nabla \phi_j + \int_{\Omega} \rho \left( V_h \cdot \nabla \phi_j \right) \cdot \phi_j d\Omega = 0 \quad (12)$$

The tangential and normal components  $u_t$  and  $u_n$  are taken as  $\phi_j \cdot \vec{t}$  and  $\phi_j \cdot \vec{n}$  respectively using conventional notations for tangential and normal vectors to element. Finally the above discretized set of equations are written as a system of nonlinear algebraic equations as follows:

$$LU = 0, \quad (13)$$

$$SU + N(U) - L^T p = F. \quad (14)$$

Here  $U$  stands for the vector of unknowns  $v_{1j}$ ,  $v_{2j}$  and  $v_{3j}$ ,  $p$  denotes the vector of unknowns  $p_j$ ,  $N$  represents the discretized nonlinear convective terms,  $LU$  is the discretization of continuity equation and  $-L^T p$  is the discretization of pressure gradient. All the contributions from boundary integrals are transferred to right hand side. In order to handle nonlinear terms we have chosen standard Newton linearization method. So in linearized form the system is written as:

$$SU + N(U^k)U - L^T p = F, \quad (15)$$

$$LU = 0. \quad (16)$$

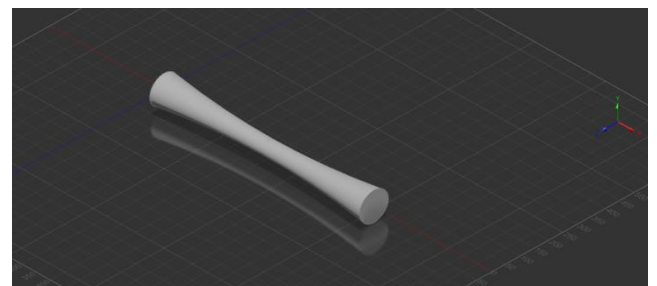
where  $U^k$  refers to the solution from previous iteration.

## Numerical Implementation

The implementation for the simulation of model presented in section II are carried within the framework of open source Autodesk. In order to achieve accurate and stable computations of gradients a second order upwinding scheme is used for momentum equations.

### Simulation setup

The formulation presented in section II is implemented in a three dimensional converging-diverging nozzle. Figure 1 shows the simulation domain. The nozzle is aligned along x-axis and three dimensional flow is considered. A finite velocity magnitude is provided at the inlet and velocity constant is kept unknown at outlet with a fixed pressure value. No-slip boundary are used for the symmetry walls of channel.



**Figure 1:** Schematic diagram of converging-diverging nozzle aligned along x-axis.



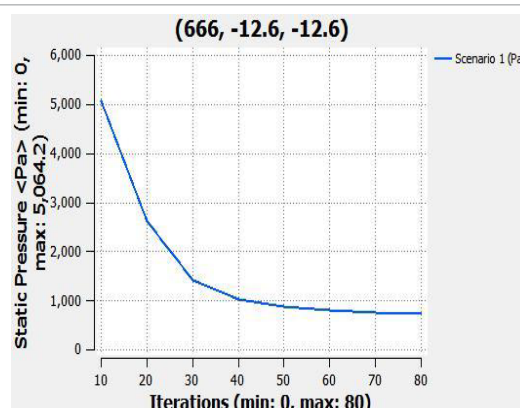
**Figure 2:** X-component of velocity vector.



**Figure 3:** Y-component of velocity vector.



**Figure 4:** Z-component of velocity vector.



**Figure 5:** XY Plot for Static Pressure.

## Simulation Results

Figures 1-8 give the overview of different flow parameters. These are velocity components in each direction and shear stresses. The power law non-Newtonian model is used with viscosity coefficient equals to 0.0033. The mass flow rate of 5 g/s is applied at the inlet. The flow is turbulent with k-epsilon model and Kappa taken as 0.4.

## Conclusion

The steady turbulent second grade fluid flow has been observed by computational method. The flow regime has been bounded by a converging-diverging nozzle. It is observed that flow is fully three-dimensional near converging throat area inlet. But this effect reduces near inlet and outlet boundaries. Moreover shear stresses are maximum

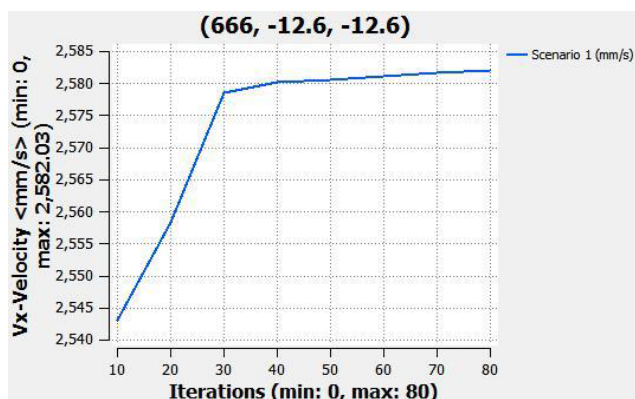


Figure 6: XY plot for x-component of velocity vector.

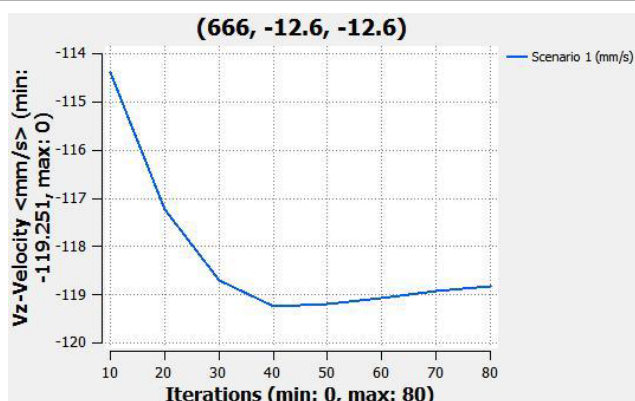


Figure 7: XY plot for z-component of velocity vector.

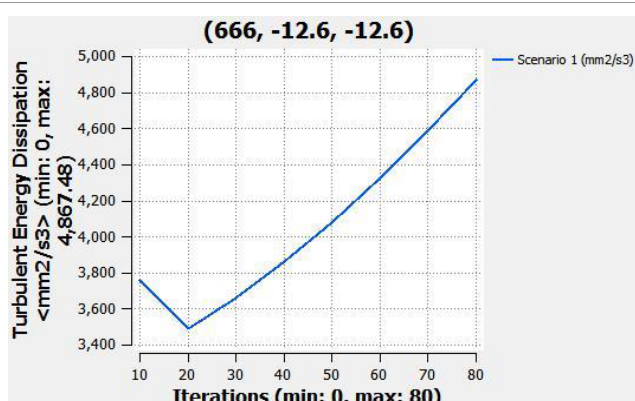


Figure 8: XY plot for turbulence kinetic energy dissipation.

in converging throat area. There is a significant decrease in pressure in diverging regime while turbulence kinetic energy dissipation effects are maximum for this region. Hence it can be inferred that there is a relation between pressure loss and mass flow rate.

The future goals are to extend the study for multi physics problems by considering the thermal and magnetic effects. Moreover the nozzle walls can be designed as wavy or irregular to optimize the flow parameters.

#### Acknowledgments

Sidrah Ahmed and Zarqa Bano acknowledge the financial and technological support from Sukkur Institute of Business Administration for this work.

#### References

- Jeffery GB (1915) L The two-dimensional steady motion of a viscous fluid. The London, Edinburgh, and Dublin Philosophical Magazine and Journal of Science 29: 455-465.
- Hamel G (1917) Spiralförmige Bewegungen zäher Flüssigkeiten. Jahresbericht der Deutschen Mathematiker-Vereinigung 25: 34-60.
- Schlichting H, Gersten K (2003) Boundary-layer theory. Springer Science and Business Media.
- Batchelor GK (2000) An introduction to fluid dynamics. Cambridge university press.
- Makinde OD, Sibanda P (2000) Steady flow in a diverging symmetrical channel: numerical study of bifurcation by analytic continuation. Quaestiones Mathematicae 23: 45-57.
- Siddiqui AM, Haroon T, Bano Z (2014) Steady 2-D Flow of a Second Grade Fluid in a Symmetrical Diverging Channel of Varying Width. Applied Mathematical Sciences 8: 4675-469.
- Mehta RC, Suresh K, Iyer RN (1998) Thermal stress analysis of a solid rocket motor nozzle throat insert using finite element method. Indian journal of engineering and materials sciences 5: 271-277.
- Jagtap R (2014) Theoretical & CFD analysis of de laval nozzle. International Journal of Mechanical and Production Engineering 2: 33-36.
- Sanjay RP, Subba Rao DV (2015) Design and Analysis of Hot Runner Nozzle Using Fem. International of Mechanical and Industrial Technology 3: 228-241.
- Walters K (1970) Relation between coleman-noll, rivlin-ericksen, green-rivlin and oldroyd fluids. Zeitschrift für angewandte Mathematik und Physik ZAMP 21: 592-600.
- Sequeira A, Baía M (1999) A finite element approximation for the steady solution of a second-grade fluid model. Journal of Computational and Applied Mathematics.