A numerical simulation for solving two dimensional non-Newtonian jet buckling in Moulding Process by VOF method

M. Passandideh Fard , B. Bagherian
Mechanical department
Ferdowsi university of mashhad
Mashhad, Vakil abad BLV, Azadi square
IRAN
behtash60@gmail.com

Abstract: - This work presents a numerical technique for solving two-dimensional non- Newtonian free surface flows. The governing equations are solved by VOF method on a staggered grid. It uses marker particles to describe the fluid. To predict the shape of the jet buckling, Navier-Stokes equations are used while The viscosity ($\upsilon(\dot{\gamma})$) as a function of shear rate $(\dot{\gamma})$ which determined by (Δ :Δ). A modified Volume-of-Fluid (VOF) technique based on Youngs' algorithm is used to track the jet buckling. As currently implemented, the present method can simulate two-dimensional non Newtonian flow in which the viscosity is modeled by using the Cross model. Buckling is a physical instability; it is the fluid equivalent of Euler's slender rod subject to a compressive force .Like the Euler's rod, the fluid may buckle one way or the other and indeed for a 2D fluid jet it may buckle in any direction. the simulation of jet buckling is approved by both experimental and theoretical results of Cruickshank and Munson [7] and Cruickshank [8]. this concurs with results obtained by Tom´e et al. [9] for the two-dimensional case.

Key-Words: - non Newtonian fluid- two dimensional- free surface flows- VOF method- jet buckling
Cross model

1 Introduction

Non-Newtonian fluid flows with free surfaces become visible in many industrial processes: injection moulding (plastic industries), container filling (food industry), inkjet devices, wire coating, among others, are all examples of non-Newtonian free surface flows problems The early days (1975–1985) of computational rheology are well covered by the reference works of Crochet et al.[1] and Keunings [2]. More recent reviews include those of Keunings [3] and Owens and Phillips [4]. 35]). by regardig the polymeric conformation model as coarse-grained, we can employ kinetic theory models and these can be treated by stochastic simulation. The next level is the macroscopic approach of continuum mechanics. At this point, a suitable constitutive equation is required for closure of the system and this normally leads to a system of partial differential equations. However, integral constitutive equations are becoming far more prominent (since they may be directly derived from molecular theories (see,e.g. Doi and Edwards [6])), leading to systems of partial integro-differential equations; a recent comprehensive review on integral viscoelastic fluids is that by Keunings [3]. Obviously, in principle continuum models may be derived from kinetic theory. However, only the linear dumbell model leads to a constitutive relation from which the Oldroyd B model is obtained. In addition to continuum modeling, two other approaches are finding favor: dissipative particle dynamics.and lattice Boltzmann models.The partial differential equations or partial integro differential equations are then solved by numerical methods. principally finite element but also by finite differences, finite volumes and spectral methods (e.g. [4]). Computational rheology involving time-dependent free surfaces is less common, although many of the Lagrangian approaches for confined flows may at least in principle be used to solve free surface and interfacial flows. The present paper solves for the flow and transient non-Newtonian fluid on a highly irregular two dimensional region with multiple free surfaces, and with the viscosity depending in a prescribed way upon the local shear rate. The key features of the method proposed are the accurate approximation. The purpose of the paper is to describe the numerical algorithm in just sufficient detail to make it reproducible and, at the same time, display the results of the application of the code to jet buckling.

The dynamic behavior of two-phase flows is of great importance in various processes ranging from engineering applications to environmental phenomena. A well-known method for tracking the free surface of a

liquid is Volume-of-Fluid (VOF) technique (Passandideh-Fard and Roohi, 2008) where the computational domain is characterized by a liquid volume fraction function. This function is used to determine both the liquid position and the liquid/gas interface orientation. Roughly two important classes of VOF methods can be distinguished with respect to the representation of the interface, namely simple line interface calculation (SLIC) and piecewise linear interface calculation (PLIC). Earlier works with VOF were generally based on the SLIC algorithm introduced by Noh and Woodward (1976) and the donor-acceptor algorithm published by Hirt and Nicholas (1981). More accurate VOF techniques include the PLIC method of Youngs (1982). The accuracy and capabilities of the older VOF algorithm such as the Hirt-and-Nicholas VOF method were studied by Rudman, 1997.
Front tracking methods (Unverdi and Tryggvason, 1992; Esmaeeli and Tryggvason, 1998a, 1998b; Tryggvasson et al., 2001) make use of markers (for instance triangles), connected to a set of points, to track the interface whereas a fixed or Eulerian grid is used to solve the Navier-stokes equations. This method is extremely accurate but also rather complex to implement due to the fact that dynamic re-meshing of the Lagrangian interface mesh is required and mapping of the Lagrangian data onto the Eulerian mesh has to be carried out. Difficulties arise when multiple interfaces interact where all require a proper sub-grid model. Contrary to most other methods, the automatic merging of interfaces does not occur in front tracking techniques due to the fact that a separate mesh is used to track the interface. In this study, full two-dimensional shear-thinning multiple
free surface flows are discussed. The model is the generalized Newtonian fluid with the viscosity related to the shear rate by the Cross model, by a modified Volume-of-Fluid (VOF) technique based on Youngs' algorithm.

2 Problem Formulation

In this section, we present a brief account of the numerical method. The flow governing equations are:

$$\vec{\nabla}\cdot\vec{V} = 0 \qquad (1)$$

$$\frac{\partial \vec{V}}{\partial t} + \vec{\nabla}\cdot(\vec{V}\vec{V}) = -\frac{1}{\rho}\vec{\nabla}p + \frac{1}{\rho}\vec{\nabla}\cdot\vec{\tau} + \frac{1}{\rho}\vec{F}_b \qquad (2)$$

where $\vec{V}$ is the velocity vector, p is the pressure and $\vec{F}_b$ represents body forces acting on the fluid. Using VOF method by means of a scalar field f whose value is unity in the liquid phase and zero in the gas. When a cell is partially filled with liquid, f will have a value between zero and one.

$$f = \begin{cases} 1 & \text{in liquid} \\ > 0,\ < 1 & \text{at the liquid-gas interface} \\ 0 & \text{in gas} \end{cases} \qquad (3)$$

The discontinuity in f is propagating through the computational domain according to:

$$\frac{df}{dt} = \frac{\partial f}{\partial t} + \vec{V}\cdot\vec{\nabla}f = 0 \quad \Rightarrow \quad \left(\frac{\partial f}{\partial t}\right)_{exact} = -(\vec{V}.\vec{\nabla})\, f \quad (4)$$

Although the velocity field is divergence free, the term $(\vec{\nabla}\cdot\vec{V})$ has an order of $O(\varepsilon)$ in numerical solution. Therefore, in order to increase the accuracy of the numerical solution, Eq. 4 is used in the conservative form as

$$\left(\frac{\partial f}{\partial t}\right)_{numerical} = -(\vec{V}.\vec{\nabla})\, f - (\vec{\nabla}\cdot\vec{V})\, f = -\vec{\nabla}\cdot(\vec{V}f) \quad (5)$$

where

$$\left(\frac{\partial f}{\partial t}\right)_{exact} = \left(\frac{\partial f}{\partial t}\right)_{numerical} + (\vec{\nabla}\cdot\vec{V})\, f \qquad (6)$$

For the advection of volume fraction f based on Eq. 4, different methods have been developed such as SLIC, Hirt-Nichols and Youngs' PLIC . The reported literature on the simulation of free-surface flows reveals that Hirt-Nichols method has been used by many researchers. In this study, however, we used Youngs' method which is a more accurate technique. Assuming the initial distribution of f to be given, velocity and pressure are calculated in each time step by the following procedure.

$$\tilde{f} = f^{\,n} - \delta t\vec{\nabla}\cdot(\vec{V}f^{\,n}) \qquad (7)$$

Then it is completed with a "divergence correction"

$$f^{\,n+1} = \tilde{f} + \delta t\,(\vec{\nabla}\cdot\vec{V})\, f^{\,n} \qquad (8)$$

A single set of equations is solved for both phases, therefore, density and viscosity of the mixture are calculated according to:

$$\begin{aligned} \rho &= f\rho_l + (1-f)\rho_g \\ \mu &= f\mu_l + (1-f)\mu_g \end{aligned} \qquad (9)$$

where subscripts l and g denote the liquid and gas, respectively. New velocity field is calculated according to the two-step time projection method as follows. First, an intermediate velocity is obtained,

$$\frac{\tilde{\vec{V}} - \vec{V}^n}{\delta t} = -\vec{\nabla}\cdot(\vec{V}\vec{V})^n + \frac{1}{\rho^n}\vec{\nabla}\cdot\vec{\tau}^n + \vec{g}^n + \frac{1}{\rho^n}\vec{F}_b^{\,n} \qquad (10)$$

The continuum surface force (CSF) method is used to model surface tension as a body force ($\vec{F}_b$) that acts only on interfacial cells. Pressure Poisson equation is then solved to obtain the pressure field,

$$\vec{\nabla}\cdot[\frac{1}{\rho^{n}}\vec{\nabla}p^{n+1}]=\frac{\vec{\nabla}\cdot\tilde{\vec{V}}}{\delta t} \quad (11)$$

Next, new time velocities are calculated by considering the pressure field implicitly,

$$\frac{\vec{V}^{n+1}-\tilde{\vec{V}}}{\delta t}=-\frac{1}{\rho^{n}}\vec{\nabla}p^{n+1} \quad (12)$$

The local shear rate is given by

$$\dot{\gamma}=\Delta:\Delta \quad (13)$$

In two-Dimensional study $\dot{\gamma}$ becomes

$$\dot{\gamma}=\left[2\left(\frac{\partial u}{\partial x}\right)^{2}+2\left(\frac{\partial v}{\partial y}\right)^{2}+\left(\frac{\partial u}{\partial y}+\frac{\partial v}{\partial x}\right)^{2}\right] \quad (14)$$

The viscosity $\upsilon(\dot{\gamma})$ can be any function of $\dot{\gamma}$ representing the shear-thinning nature of the fluid. In the simulations presented in this paper, we employed the Cross mode

$$\upsilon(\dot{\gamma})=\upsilon_{\infty}+\frac{\upsilon_{\circ}-\upsilon_{\infty}}{\left(1+(K\dot{\gamma})^{m}\right)} \quad (15)$$

where m, $\upsilon_{\circ}$, υ_{∞} and K are given positive constants.

3 Problem Solution

When a low Reynolds number jet flows onto a rigid plate, a phenomenon known as jet buckling can occur if the Reynolds number is smaller than a prescribed value. This phenomenon has attracted a number of investigators and it has been studied both experimentally and numerically. Cruickshank and Munson [7] and Cruickshank [8] have presented both experimental and theoretical results for Newtonian jets. From their study, they obtained estimates for when jet buckling might occur: these are based upon the Reynolds number and the ratio *H/D* (*H* is the height of the inlet to the rigid plate and *D* is the jet diameter). In particular, from a one-dimensional stability study, they found that a two-dimensional jet will buckle if the following conditions

$Re < .5$ and $H/D > 10$

are satisfied. For an ax symmetric jet, they found that the buckling conditions were modified to

$Re < 1.2$ and $H/D > 7$

we present in this section two calculations that simulate the buckling of an ax symmetric viscous jet hitting a rigid plate. For a Newtonian jet, the Reynolds number dictates the behavior of the flow, whereas for a generalized Newtonian jet the "Reynolds number" We consider a flat surface and an ax symmetric jet issuing from an ax symmetric nozzle onto the flat surface at a prescribed velocity (see Fig. 1). The following input data were employed: domain dimensions, 10 mm × 30 mm; mesh size, 62 × 102 cells ($\delta_x \neq \delta_y$); jet diameter (*D*) = 2.5 mm; fluid velocity at the nozzle (*u*) = $0.1\, ms^{-1}$;

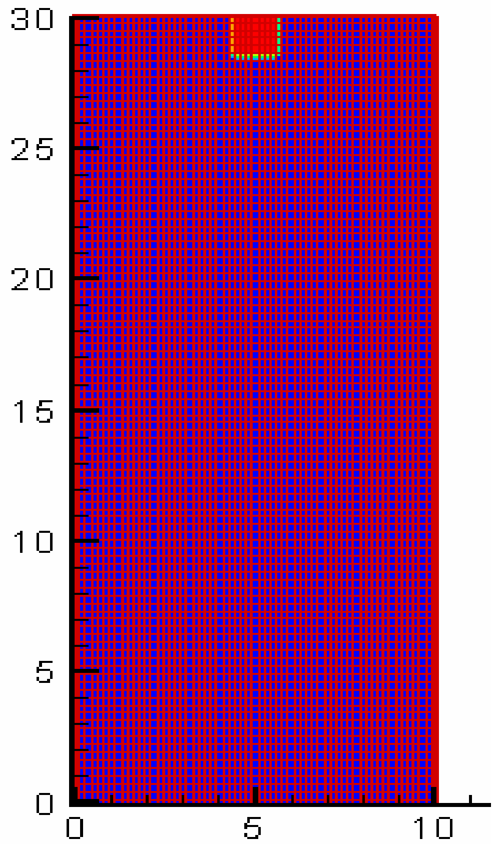

Fig. 1. Numerical simulation and meshing of a non- Newtonian jet onto a flat surface at times $t = 0$ s

flat surface dimensions, 10 mm×30 mm ; nozzle dimensions, 2.5 mm diameter and 2mm height; height of nozzle (*H*) = 28 mm (distance of the nozzle to the flat surface); gravity was taken to act in the negative *y*-direction with $g_y = -9.81\, ms^{-2}$. The fluid properties were chosen to be

υ_{∞}=0.006 $m^2 s^{-1}$, $\upsilon_{\circ}$=0.0006 $m^2 s^{-1}$, K=.3, m=.4

The inlet diameter is *D* = 3mm and the inlet velocity is *U* = 0.1 ms^{-1}. This gives $Re = UD/\upsilon_{\circ} = 0.4$ and a slenderness ratio of *H/D* = 11. Again, Cruickshank's analysis predicts that the non Newtonian jet will buckle.

As it is seen we show all domain in Fig. 2,3. It is obvious that the non Newtonian jet becomes thick on hitting the rigid plate and then at time *t* = 1.4050e-1 s, it buckles whereas the two- dimensional Newtonian jet does not become noticeably thicker but flows radially without any evidence of buckling.

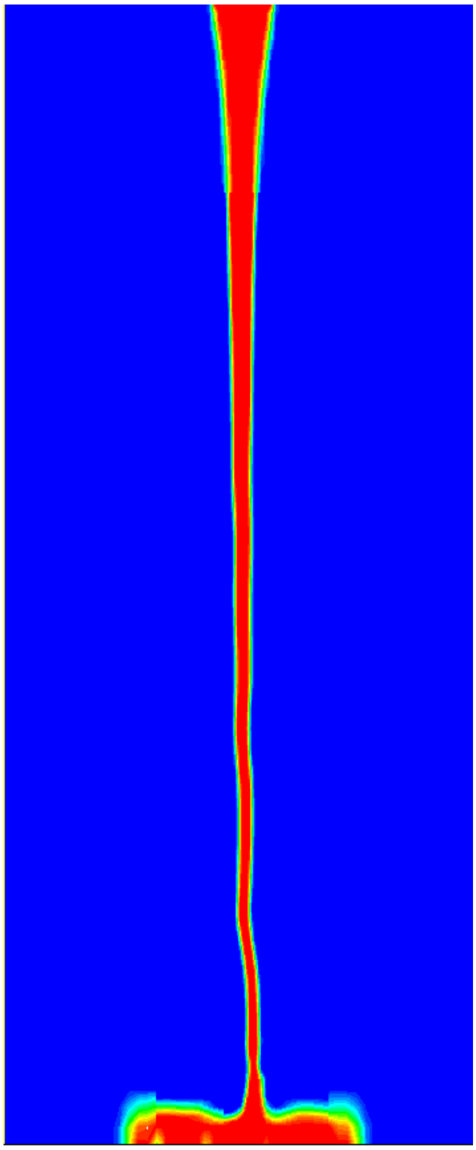

Fig. 2. Numerical simulation of two-dimensional non Newtonian jet buckling at t=1.4050e-1 s

Indeed, as the flow is hindered by the no-slip condition on the rigid plate, the non Newtonian fluid accumulates, and by obstructing the oncoming fluid initiates buckling. as it is exposed in Fig.4, Fig.5 we only show the region that we guess buckling happened at t= 1.5216e-1 s. It is observable that in non-Newtonian case jet becomes thick but in Newtonian jet we can see the jet dos not become thick from middle of the jet. Fig.6, Fig.7 also show the same result at t=1.7591e-1 s.Fig.8, Fig.9 are presenting the
identical results at t=2.8791e-1 s, that non-Newtonian fluid buckles at different times while Newtonian fluid dose not buckle
.

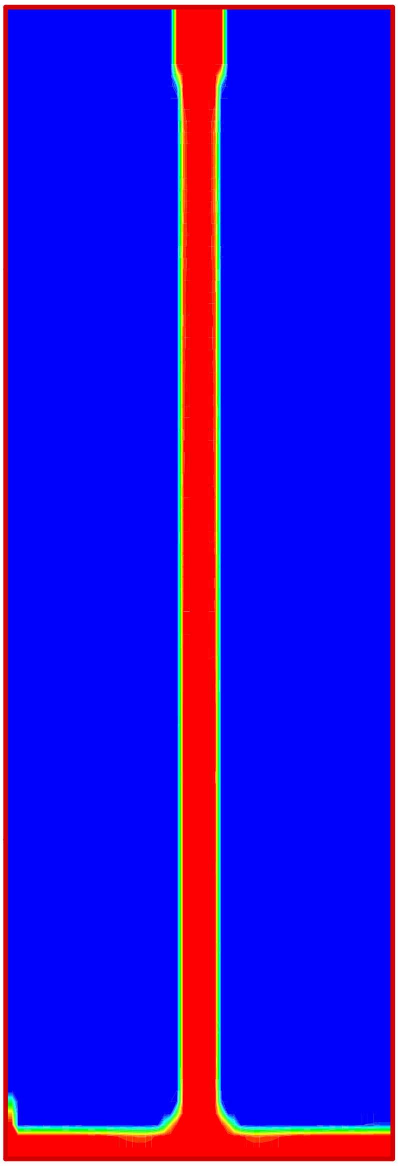

Fig. 3. Numerical simulation of two-dimensional Newtonian jet at t=1.4050e-1 s

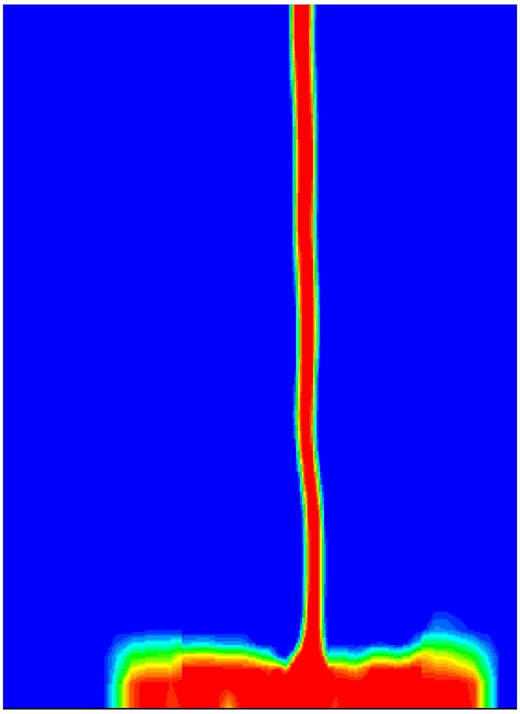

Fig. 4. Numerical simulation
of two-dimensional non Newtonian
jet buckling at t=1.5216e-1s

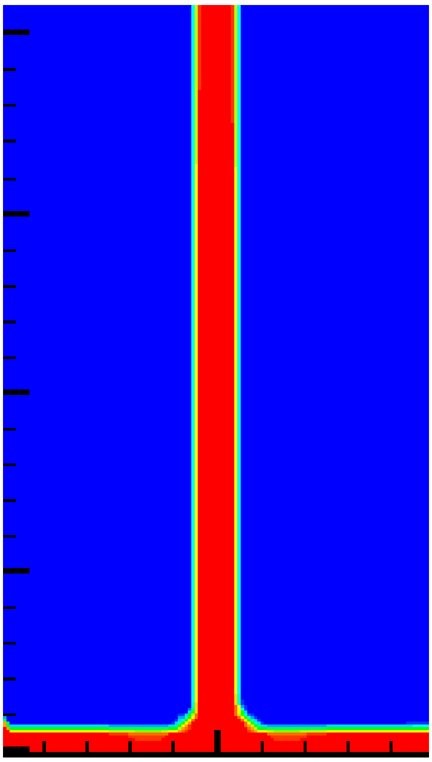

Fig. 5. Numerical simulation
of two-dimensional
Newtonian jet at t=1.5216e-1s

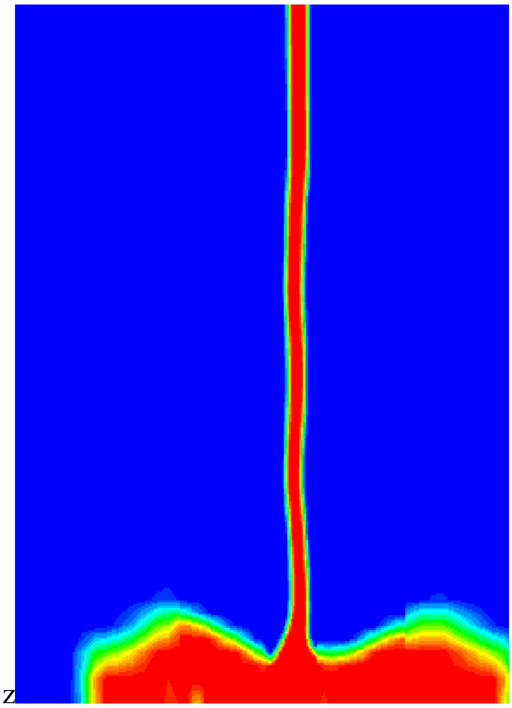

Fig.6. Numerical simulation
of two-dimensional non Newtonian
jet buckling at t=1.7591e-1s

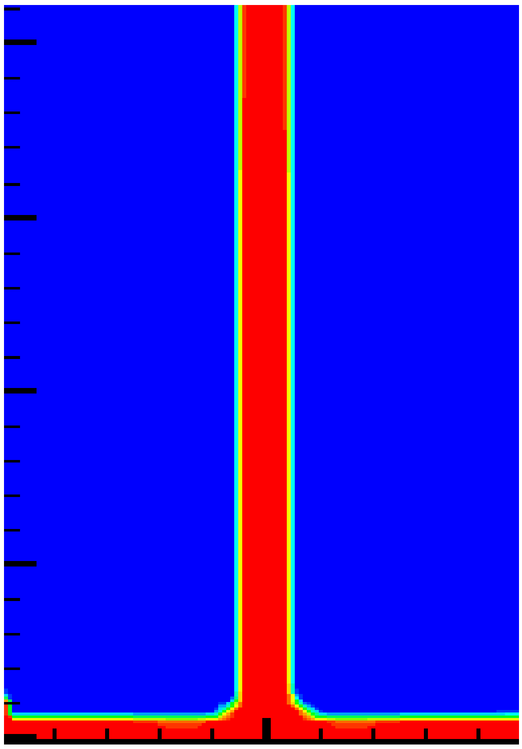

Fig.7. Numerical simulation
of two-dimensional
newtonian jet at t=1.7591e-1s

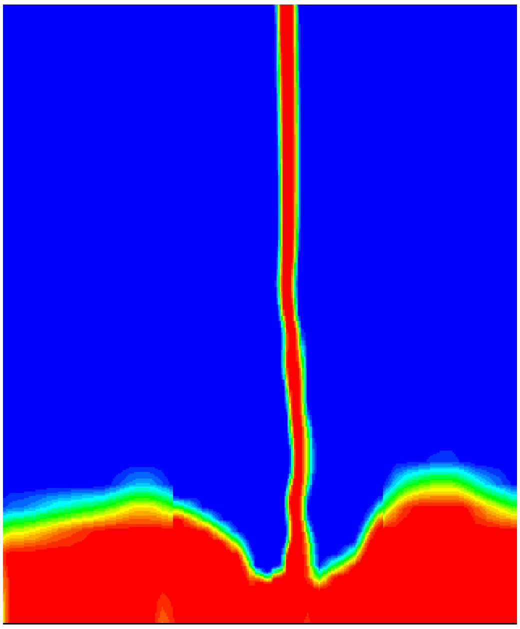

Fig. 8. Numerical simulation of two-dimensional non Newtonian jet buckling at at t=2.8791e-1s

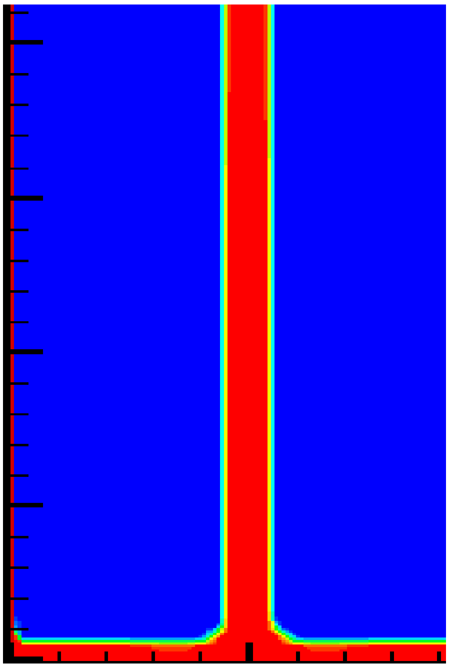

Fig. 9. Numerical simulation of two-dimensional newtonian jet at t=2.8791e-1s

Buckling is a physical instability; it is the fluid equivalent of Euler's slender rod subject to a compressive force .Like the Euler's rod, the fluid may buckle one way or the other and indeed for a 2D fluid jet it may buckle in any direction. Of course, a small perturbation is required for the fluid to buckle.

4 Conclusion

This paper presented VOF technique for solving two-dimensional non Newtonian free surface flows. In this work, attention has been given to the implementation of the 2D-free surface conditions. The finite difference equations presented have been implemented to produce a two-dimensional code. jet buckling simulation clearly displayed the behavior of shear-thinning fluids. the simulation of jet buckling is approved by both experimental and theoretical results of Cruickshank and Munson [18] and Cruickshank [19]. the simulation of jet buckling showed that under certain circumstances a non Newtonian jet may buckle while a generalized Newtonian jet may not; this concurs with results obtained by Tom´e et al. [20] for the two-dimensional case.

References:

[1] M.J. Crochet, A.R. Davies, K.Walters, Numerical Simulation of Non-Newtonian Flow, Elsevier, 1984.

[2] R. Keunings, Simulation of viscoelastic fluid flow, in: C.L.Tucker (Ed.), Fundamentals of Computer Modeling for PolymerProcessing, Hauser Publishers, Munich, 1989, pp. 404–469.

[3] R. Keunings, Finite element methods for integral viscoelastic fluids,in: Keynote Lecture at the Eighth International Symposium on Computational Fluid Dynamics, Bremen, Germany, 5–10 September 1999.

[4] R.G. Owens, T.N. Phillips, Computational Rheology, Imperial College Press, London, 2002.

[5] C. Aust, M. Kroger, S. Hess, Structure and dynamics of dilute polymer solutions under shear flow via nonequilibrium molecular dynamics, Macromolecules 32 (1999) 5660–5672.

[6] M. Doi, S.F. Edwards, The Theory of Polymer Dynamics, Clarendon Press, Oxford, 1986.

[7] J.O. Cruickshank, B.R. Munson, Viscous-fluid buckling of plane andaxisymmetric jets, J. Fluid Mech. 113 (1981) 111–127.

[8] J.O. Cruickshank, Low-Reynolds-number instabilities in stagnating jet flows, J. Fluid Mech. 193 (1988) 111–127.

[9] M.F. Tom´e, B. Duffy, S. McKee, A numerical technique for solvingunsteady non-Newtonian free surface flows, J. Non-Newtonian Fluid