Numerical Simulation of a Non Newtonian Free Surface Flow by VOF Method in a Wire Coating Process

Mohammad P. Fard[1], Behtash Bagherian [2]

[1] Assistant Professor, Ferdowsi University of Mashhad; mpfard@um.ac.ir
[2] Graduate Student, Ferdowsi University of Mashhad; behtashb60@gmail.com

Abstract
Wire coating is a process which includes electronic wires or any kind of protecting wires by coating them with another material. In this paper we simulated Wire coating process which used in welding industry for making electrods. Simulation is based on Volume-of-Fluid (VOF) method where transient 2D Navier-Stokes equations by using cross model for non-Newtonian fluid are solved. In this process non-Newtonian fluid extrudes on wire. Thickness of non-Newtonian fluid and pressure fluctuation are compared with their available experiment.

Keywords: Wire coating, volume of fluid, Cross model, non-Newtonian fluid

Introduction

Coated welding electrod was first desighned by Oscar Kjellberg. He used a mixture of organic and mineral materials with a binder as a coating layer for electerod. This coated layer helped electrod to resist against combination with surrounding. This was a revolution in welding industry and changed the world.

Though in spite of extensive amount of using coated welding electrod, there are few studies about fluid behavior of coated layer.

Orlov [1] developed a double die arrangement using externally pressurized dough. The lubricant was transported into the chamber formed by the exit cone of the pressure die and the entry part of the drawing die, where the pressurized lubricant provides the hydrodynamic lubrication during drawing. Though better result was claimed for reduced die wear and reduced power consumption, there was a lack of substantial evidence to support this claim. Later, Thompson and Symmons [2] experimented with polymer melt as a means of dough. Crampton further investigated plasto-hydrodynamic polymer lubrication of wire. He showed in his analysis that the reduction and coating thickness on the wire depends on the wire speed, polymer melt viscosity and the die chamber configuration.

This was further investigated by Parvinmehr [3] who used a die-less unit and the smallest bore of which was slightly greater than the diameter of the wire. For both of these works, the tests were carried out with wires of 2 mm diameter. They also carried out a non-Newtonian analysis for wire drawing using a stepped parallel bore unit. Later, Hashmi[4] reported, a non Newtonian analysis for wire drawing in a conical tubular orifice. For plasto-hydrodynamic pressure through a combined parallel and tapered bore unit for wire coating process only without any reduction in wire diameter. In this paper, a non-Newtonian wire coating model has been presented, the solution of which gives the prediction of the coating thickness and the pressure distribution within the unit.

System of Units

SI system of units is deemed to be used. If necessary use the equivalent value in the other system of units in brackets after the SI system of units.

Direct Problem

Consider a two-dimensional axisymmetric 60*30 millimeter domain with a mesh size of 120*60 nodes. We have a wire with which comes out of a store full of a high viscose non Newtonian fluid. Non Newtonian fluid coats the wire. We have inflow condition at top and bottom and high pressure at left and right sides of store.

Equations

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

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

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

Where $\vec{V}$ is the velocity vector, p is the pressure, ρ is the density and $\vec{F_b}$ represents body forces acting on the fluid. 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:

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

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

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

Where

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

For the advection of volume fraction f based on Eq. 4, different methods have been developed such as SLIC, Hirt-Nichols and Youngs' PLIC. In this study, we used Youngs' method [4], which is a more accurate technique.

The f advection begins by defining an intermediate value of f,

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

Then it is completed with a "divergence correction"

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

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

$$\begin{aligned}\rho &= f\rho_L + (1-f)\rho_G \\ \mu &= f\mu_L(\dot{\gamma}) + (1-f)\mu_G\end{aligned}, \quad (9)$$

Where subscripts L and G denote the liquid and gas,

$$\dot{\gamma} = ((\frac{\partial u}{\partial x})^2 + (\frac{\partial v}{\partial y})^2 + (\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x})^2)^{0.5}$$

(10)

And $\mu(\dot{\gamma})$ in two dimensional cross method calculated according to:

$$\mu(\dot{\gamma}) = \frac{\mu_0}{(1+k(\dot{\gamma})^m)}$$

(11)

Where m and k are cross numbers. These numbers are empirical.

New velocity field is calculated according to the two step time projection method as follows. First, an intermediate velocity is obtained,

$$\frac{\tilde{\vec{V}} - \vec{V}^n}{\delta t} = -\vec{\nabla}\cdot(\vec{V}\vec{V})^n + \frac{1}{\rho^n}\vec{\nabla}\cdot\vec{\tau}^n + \vec{g}^n + \frac{1}{\rho^n}\vec{F}_b^n \quad (12)$$

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,

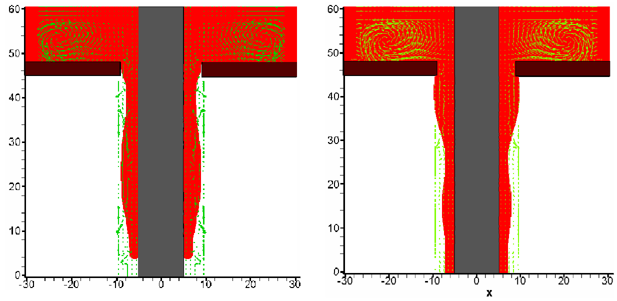
$$\vec{\nabla}\cdot[\frac{1}{\rho^n}\vec{\nabla}p^{n+1}] = \frac{\vec{\nabla}\cdot\tilde{\vec{V}}}{\delta t} \quad (13)$$

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

$$\frac{\vec{V}^{n+1} - \tilde{\vec{V}}}{\delta t} = -\frac{1}{\rho^n}\vec{\nabla}p^{n+1} \quad (14)$$

Resultsand Discussion

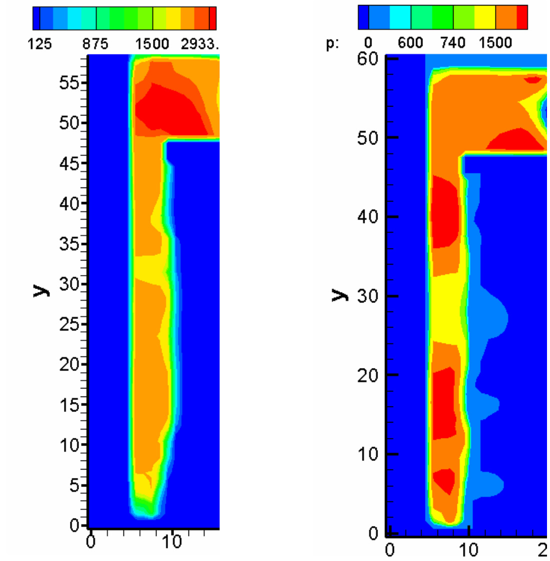
As a first step we assume Cross model for non Newtonian fluid. First we assume 1500 kPa for pressure. To validating by experiment results we choose cross numbers k=1, m=.7 and velocity=0.5 m/s. viscosity of fluid is $0.08\,m^2s^{-1}$. Figure 1 shows the phase contours and velocity vector in two different times.



t=07 s t=09s

Figure 1. phase contour and velocity vector, p=1500 kPa and v=.5 m/s

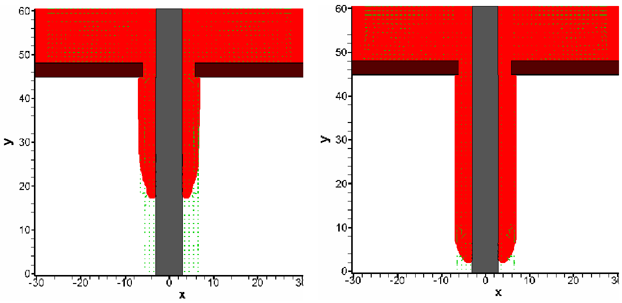
Fig 2 shows the pressure contour in the same times.



t=07 s t=09s

Fig 2. pressure contour, p=1500 kPa and v=.5 m/s

As we can see in Figure 2 pressure maintains some of its initial quantity on the rod after exiting the cylinder. This phenomenon happens because of viscoelastic memory in non Newtonian fluids [6].

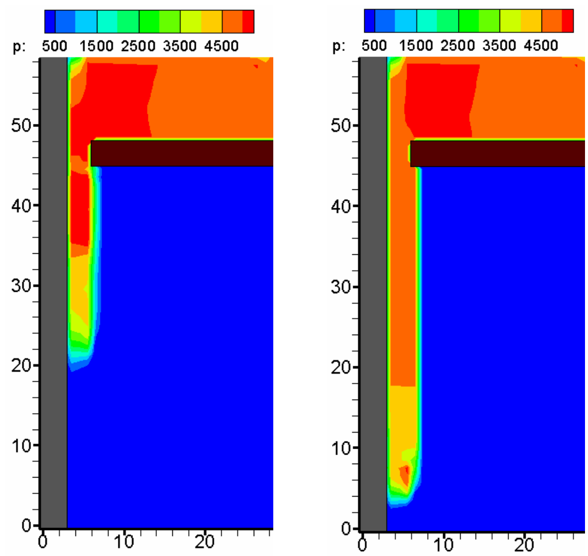


t=0.5 s t=0.73s

Figure 3. phase contour and velocity vector, p=5500 kPa and v=.16 m/s

Then we assume 5500 kPa for pressure and k=1, m=.5 for cross model and velocity=0.14 m/s for cross model to comparison with experimental data. Viscosity of fluid is 0.14 $m^2 s^{-1}$.

Figure 3 shows the phase contour and velocity vector in two different times. We can see in this figure that thickness of non Newtonian dough in this pressure is less than 1500 kPa.

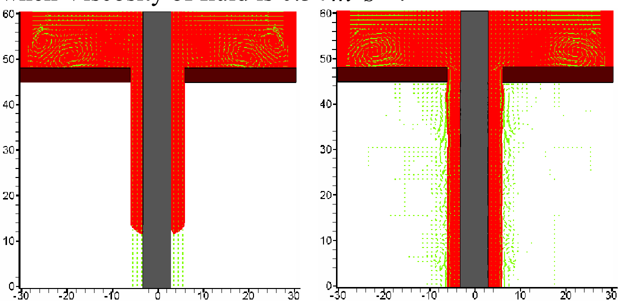


t=0.5 s t=0.73s

Figure 4. pressure contour, p=5500 kPa and v=.16 m/s

In Figure 4 pressure contour at the same time has been shown.

We also compare numerical and experimental results in p=10000 kPa with k=1 and m=0.5 and velocity=0.7 m/s. when Viscosity of fluid is 0.34 $m^2 s^{-1}$.



t=0.35 s t=054 s

Figure 5. phase contour and velocity vector, p=10000 kPa

Figure 5 and 6 shows phase and pressure contour in different times. It is shown that eddy is diminished because of high viscose fluid [7].

We can see in lower pressure and viscosity because of discrepancy between surface tension and shear tension on the wall there is a contraction, expansion behavior in fluid. In higher pressure and viscosity this behavior is negligible [8].

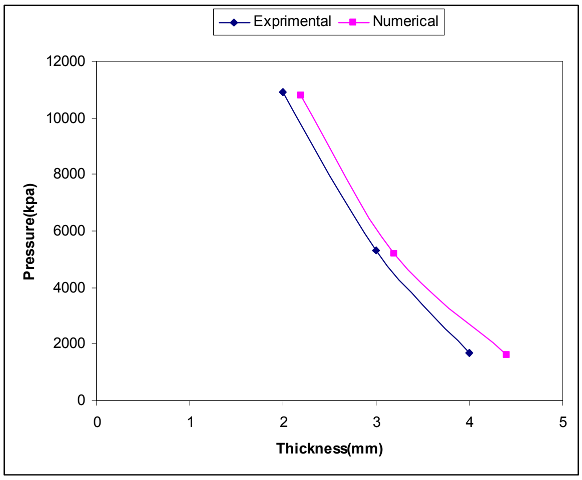


Figure 7. comparing model predictions and experimental results

In Figure 7 we compare model predictions and experimental results. We can see numerical results are agreed with experimental results.

Conclusions

This Simulation is based on Volume-of-Fluid (VOF) method where transient 2D Navier-Stokes equations by using cross model for non-Newtonian fluid are solved. We studied 3 different pressure extrusions with 3 different thicknesses on their rods. We can see higher pressure lower rod thickness. In high pressure eddy is diminished because of high viscose fluid [7]. In lower pressure and viscosity because of discrepancy between surface tension and shear tension on the wall there is a contraction, expansion behavior in fluid, when In higher pressure and viscosity this behavior is negligible [8].

List of Symbols

g	gravitational acceleration
V	velocity
P	pressure
t	time
ΔP	pressure difference
f	fractional amount of liquid
m	Cross number
k	Cross number

Greek symbols

μ	Viscosity
ρ	density
τ	Shearstress
$\dot{\gamma}$	Shear rate

References

1- S.I. Orlov, V.T. Stukalov, Integrated development and introduction of new high speed mills for drawing wires, in Steel in the USSR, 10, 1974, pp. 953–956.

2- P.J. Thompson, G.R. Symmons, A plasto-hydrodynamic analysis of coating of wire using polymer melt during drawing, Proc. Inst. Mech. Eng. 191 (1997), pp. 115-132.

3- H. Parvinmehr, G.R. Symmons, A non-Newtonian plasto-hydrodynamic analysis of die-less wire drawing process Int. J. Mech. Sci. 29 (4) (1987), pp. 239–257.

4- M.S.J. Hashmi, G.R. Symmons, A numerical solution for the plasto hydrodynamic drawing of a rigid non-linear strain hardening continuum through a conical orifice, J. Math model. 8 (1995), pp.457–462.

5- Youngs, D.L., “Time dependent multi material flow with large fluid distortion”, Numer. Methods for Fluid Dynamics, N.Y,15, (1992), pp. 273–285.

6- D.C. Christopherson, P.B. Naylor, Promotion of lubrication in Wire drawing, Proc. Inst. Mech. Eng. (1955), pp. 341-369

7- J.G. Wistrich, Lubrication in wire drawing, Inst. Mech. Eng (1957) pp. 505–511.

8- R.I. Tanner, K.Walters, Rheology: An Historical Perspective of Welding process,Elsevier, koln, 2004, pp. 159–168.