CFD SIMULATION OF NON-NEWTONIAN TWO-PHASE FLUID FLOW THROUGH A CHANNEL WITH A CAVITY

Mehdi AHMADI 1*, Ayoob KHOSRAVI FARSANI2

1 Department of Mechanical Engineering, Islamic Azad University, Shahrekord Branch, Shahrekord, Iran

2 Department of Mechanical Engineering, Islamic Azad University, Shahrekord Branch, Shahrekord, Iran

* Corresponding author; Ahmadi.m@iaushk.ac.ir

Abstract: In this paper, the numerical solution of non-Newtonian two-phase fluid flow through a channel with a cavity was studied. Carreau-Yasuda non-Newtonian model which represents well the dependence of stress on shear rate was used and the effect of n index of the model and the effect of input Reynolds on the attribution of flow were considered. Governing equations were discretized using the finite volume method (FVM) on staggered grid and the form of allocating flow parameters on staggered grid is based on marker and cell method. QUICK scheme is employed for the convection terms in the momentum equations, also the convection term is discretized by using the hybrid upwind-central scheme. In order to increase the accuracy of making discrete, second order Van Leer accuracy method was used. For mixed solution of velocity-pressure field SIMPLEC algorithm was used and for pressure correction equation iteratively line-by-line TDMA solution procedure and the Strongly Implicit Procedure (SIP) was used. As the results show, by increasing Re, the time of sweeping the non-Newtonian fluid inside the cavity decreases, the velocity of Newtonian fluid increases and the pressure decreases. In the second section, by increasing n index, the velocity increases and the volume fraction of non-Newtonian fluid after cavity increases and by increasing velocity, the pressure decreases. Also changes in the velocity, pressure and volume fraction of fluids inside the channel and cavity are more sensible to changing the Re instead of changing n index.

Key words: Non-Newtonian Fluid, Two-Phase Flow, Carreau-Yasuda Model, VOF Method, Numerical Simulation, Finite Volume Method

1. Introduction

The difference between both classes of fluids can be observed in a variety of situations. However, as far as we know, the studies on gas and non-Newtonian liquid two-phase flows across singularities in mini channel are still limited. Non-Newtonian fluids, such as paint, milk, toothpaste, animal blood, etc. are ubiquitous in nature, and they are often encountered in many industries, like in chemical engineering, plastic manufacturing, wire and fiber coating, transpiration cooling, heat pipes,
biological fluids, gaseous diffusion, drilling muds, bio-chemical engineering, food processing, oil exploration, and medical engineering [1,2]. Particularly in industrial applications, non-Newtonian fluids are of great interest, since many used fluids, lacquers or polymer solutions show non-Newtonian effects. So the viscoelastic fluid can enhance the displacement efficiency, which has been proved by lots of experimental results [3-5].

The Carreau [6] viscosity model is considered as an appropriate viscosity model to describe the rheology of the non-Newtonian liquid due to its capability to describe the zero-shear-rate and the infinity-shear-rate Newtonian viscosity limits. [7,8] presented the calculation of viscoelastic flows of Oldroyd-B fluid. They used numerical method. In another research, Huang et al. [9] studied two-dimensional incompressible fluid flow of shear cavity. Both in Euler frame and Lagrangian frame, they were simulated by SPH, FPM, the original KGF-SPH and improved KGF-SPH. Xu et al. [10] applied a fractional flow model to describe a time-variant behavior of non-Newtonian substances. Freydier et al. [11] reported high-resolution measurements of the velocity field within the front of free-surface viscoplastic surges in an inclined channel.

Over the past decades, significant advances have been made in numerical modeling of two-phase flows. For examples, the two-fluid model [12,13] and the discrete particle model [14,15] were developed to simulate two-phase flows. The front-capturing method [16] and the front-tracking [17] methods, were developed to capture the free surfaces of multi-phase flows. The flows of two immiscible liquids are encountered in a diverse range of processes and equipment. An overview of these approaches is described in [18-22].

Despite the fact that a plethora of multiphase flow models are available in the literature, the information regarding the simulation of two-phase with a non-Newtonian fluid on a channel with a cavity is quite limited. Also, despite of their importance, liquid-liquid flows have not been explored to the same extent as gas-liquid flows. In fact, gas-liquid systems represent a very particular extreme of two-fluid systems characterized by low-density ratio and low viscosity ratio. In liquid-liquid systems, the density difference between the phases is was relatively low. However, the viscosity ratio encountered extends over a range of many orders of magnitude. In the present work, the numerical solution of non-Newtonian two-phase fluid flow through a channel with cavity is studied. The input fluid in the channel is modeled as a Newtonian fluid and the liquid in the cavity is modeled as a non-Newtonian fluid. Carreau-Yasuda non-Newtonian model which represents well the dependence of stress on shear rate is used and the effect of n index of the model and the effect of input Reynolds on attribute of flow are considered. For the description of the interface, we employed the volume of fluid (VOF) method. Based on the velocity of input fluid we considered the flow laminar.

2. Modelling and simulation

2.1. Mathematical model

In this paper, non-Newtonian two-phase fluid flow in a channel with cavity was studied as shown in Fig.1. The flow, was assumed isothermal and two-dimensional. The VOF method was used to investigate the flow of polymer solution discharged into the water from the cavity. The polymer solution is modeled as a non-Newtonian fluid, and the water is modeled as a Newtonian fluid. The Carreau [6] viscosity model is considered as an appropriate viscosity model to describe the rheology of the non-Newtonian liquid, due to its capability to describe the zero-shear-rate and the infinity-shear-rate Newtonian viscosity limits. The aim of this paper is to study the effect of changes in input Re by
changing the input velocity and changes in n index of Carreau-Yasuda model. Therefore first, the flow field consist of non-dimensional velocity, pressure drop and then volume fraction of phases will be studied. To achieve this purpose in two-phase flow problems, mass fraction of two different phases in the flow field needs to be calculated. In the present study a volume of fluid (VOF) method is used. The volume fraction of a fluid is introduced as an additional variable in the governing equations.

The basic equations were discretized using the finite volume method (FVM). In the FVM, the computational domain is divided into a large number of control volumes. The primitive variables are defined at the center points of each control volume. Further information about FVM is explained in the literature [23–25]. The numerical solution of the question was conducted by SIMPLEC algorithm and computer programming was written by FORTRAN language.

2.1.1 The equations of non-Newtonian two-phase fluid

Two-phase non-Newtonian flow field in a channel with cavity is derived by continuity, momentum and a model which describes viscosity of non-Newtonian fluid. Also we need an extra equation that describe interface of two phases. The two-phase flow temperature is assumed to be constant at 300K. We consider steady, incompressible and isothermal flow in a two-dimensional Cartesian coordinate system(x,y). The above mentioned equations can be expressed as follows [26,27]:

\[ \text{div } u = 0 \]  
\[ \frac{\partial}{\partial t} (\rho u) + \rho(u, \nabla)u = \text{div } S - \nabla p \]

Where \( \rho \) is density, \( t \) is time, \( u \) is velocity and \( S \) is stress tensor that can be expressed as:

\[ S = 2\mu(\dot{\gamma})D \]  
\[ D = \frac{1}{2}[\nabla u + (\nabla u)^T] \]  
\[ \dot{\gamma} = \sqrt{2} tr(D^2) \]

\( D \) is deformation tensor and \( \dot{\gamma} \) is shear rate. For the viscosity function, we recall the Carreau model [6]:

\[ \mu(\dot{\gamma}) = \mu_\infty + (\mu_0 - \mu_\infty)[1 + (\lambda \dot{\gamma})^2]^{(n-1)/2} \]

Here, \( \mu_0 \) is zero-shear-rate viscosity, \( \mu_\infty \) is infinite-shear-rate viscosity and \( n \) are constants which are determined by experimental investigations and characterize the used fluid.

2.1.2 The Volume Of Fluid Method

The VOF method, which originates from the idea of Hirt et al. [28], is a technique where the volume fraction of each phase in a cell is calculated. The major advantage of the VOF method is its conservative nature. A brief review of VOF method can be found in [29-32]. The VOF method can also be modified or coupled with other techniques to improve its ability in solving multiphase flow problems [33-35]. The VOF function evolves according to the following advection equation:
\[
\frac{\partial f}{\partial t} + \nabla \cdot (uf) - f \nabla \cdot u = 0
\]

(7)

where \( u \) denotes the velocity vector that can be found from solution of the flow field. The VOF function \( f \) has a value between 0 and 1. By solving Eq.(1), the distribution of \( f \) is obtained. In a physical sense, Eq.(1) implies mass conservation of one phase in the mixture. Numerically, Eq.(1) is characterized as a hyperbolic or pure convection equation.

2.1.3 Boundary conditions

Generally, boundary conditions to solve the equations are as follows: No slip boundary conditions are assumed at the solid wall for velocity [36];

\[
u = 0, \quad \frac{\partial u}{\partial x} = 0, \quad \frac{\partial v}{\partial x} = 0, \quad \frac{\partial v}{\partial y} = 0
\]

(8)

The stress components at the wall are obtained by solving constitutive equations and by applying known velocity boundary conditions.

At the outlet the Neumann boundary conditions are imposed for the flow variables.

\[
\frac{\partial u}{\partial x} = 0, \quad \frac{\partial v}{\partial x} = 0, \quad \frac{\partial \tau_{xx}}{\partial x} = 0, \quad \frac{\partial \tau_{yy}}{\partial x} = 0, \quad \frac{\partial \tau_{xy}}{\partial x} = 0
\]

(9)

Boundary values for pressure-correction equations are determined by considering velocity boundary conditions. At inlet we know velocity profile.

2.2. Solution procedure

Equations that have been already derived, can be used to calculate flow field parameters. To conduct calculations in this study, governing equations are discretized using finite volume method [37-39] on staggered mesh. The form of allocating flow parameters on staggered mesh are based on marker and cell method. QUICK scheme is employed for the convection terms in the momentum equations. The convection term is discretized by using the hybrid upwind-central scheme. To increase the accuracy of making discrete, the second order Van Leer accuracy method is used [40]. For mixed solution of velocity-pressure field SIMPLEC [41] algorithm is used. For pressure correction equation iteratively line-by-line TDMA solution procedure and the Strongly Implicit Procedure (SIP) is used. To ensure the achievement of steady state, the appropriate under-relaxation technique is used. Besides that, due to use of low Reynolds number model to solve the velocity field, it is required that a sufficient number of grid points in the near-wall boundary layer. Special attention has thus been paid to the grid refinement, particularly for computing two-phase flow field. The solution process is reiterated until the maximum relative changes of flow variables (\( u, v, p, \tau_{xx}, \tau_{yy}, \tau_{xy} \)) are less then \( 10^{-6} \).

2.3. Grid Study

Grid refinement is one of the most important issues in non-Newtonian two-phase flow simulations. In order to acquire accurate results and capture realistic flow, especially near the walls and fluid phase boundaries that have considerable changes in some parameters. Also it is necessary to review the effect of varying the number of grid nodes on the flow parameters. For this purpose, the generalized Newtonian fluid flow is used at Re=500. As Fig.1 illustrates, the problem geometry consists of three sections; upstream of the cavity, the cavity, and downstream of the cavity represented by 1, 2, and 3, respectively. For different grids, different numbers of nodes are considered in the
horizontal and vertical directions. For each section, the names of these four grids are G1, G2, G3 and G4, respectively. Grid specifications are shown in Tab.1.

Table 1. Number of nodes for different grids.

<table>
<thead>
<tr>
<th>Grid</th>
<th>Number of nodes section 1</th>
<th>Number of nodes section 2</th>
<th>Number of nodes section 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>G1</td>
<td>3600</td>
<td>2400</td>
<td>3600</td>
</tr>
<tr>
<td>G2</td>
<td>7200</td>
<td>4800</td>
<td>7200</td>
</tr>
<tr>
<td>G3</td>
<td>14400</td>
<td>9600</td>
<td>14400</td>
</tr>
<tr>
<td>G4</td>
<td>28800</td>
<td>19200</td>
<td>28800</td>
</tr>
</tbody>
</table>

The uniform square grid G4 was considered as the reference mode. Given that the responses for this grid are precise, the responses for the coarser grid were compared with those of reference mode. The values of the relative errors of the longitudinal velocity at the center of channel outlet for generalized Newtonian fluid flow at Re=500 and n=0.7-1 are shown for three grids in Tab.2. As seen in Tab.2, the difference between the values obtained using G3 and G4 is negligible. Thus, the uniform grid G3 is selected to run the program.

Table 2. Values of relative errors of the longitudinal velocity at the center of the channel outlet at Re=500 and n= 0.7-1.

<table>
<thead>
<tr>
<th>n</th>
<th>EG1</th>
<th>EG2</th>
<th>EG3</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.28</td>
<td>0.12</td>
<td>0.03</td>
</tr>
<tr>
<td>0.9</td>
<td>0.30</td>
<td>0.11</td>
<td>0.04</td>
</tr>
<tr>
<td>0.8</td>
<td>0.26</td>
<td>0.18</td>
<td>0.02</td>
</tr>
<tr>
<td>0.7</td>
<td>0.27</td>
<td>0.14</td>
<td>0.02</td>
</tr>
</tbody>
</table>

2.4. Validation

Numerical simulation of non-Newtonian fluid flow in the desired geometry is done by grid G3. This grid is suitable in terms of both the accuracy and time of calculations. To investigate the performance of the computer code, the numerical results obtained are compared with analytical results at two state. In the first state as already mentioned, Carreau-Yasuda non-Newtonian model at n=1 is turned into a simple Newtonian fluid. With regards to the assumption of the generalized flow at channel outlet, the maximum velocity at the center of channel is 1.5. The longitudinal velocity derived from the numerical solution at the center of channel outlet for different Reynolds values are shown in Tab.3. In the second section to validate the computer code, computations are made in a smooth parallel plate channel and the results thereof are compared with available analytical solutions and experimental data. For n=0.5, 1, 1.25 and Re=500, a comparison of centerline velocity between this study and the analytical results [42] is shown in Tab.4. As illustrated, a good agreement is found.
Table 3. The longitudinal velocity at the center of channel outlet for different Re.

<table>
<thead>
<tr>
<th>Re</th>
<th>200</th>
<th>500</th>
<th>1000</th>
<th>1500</th>
</tr>
</thead>
<tbody>
<tr>
<td>u</td>
<td>1.499117</td>
<td>1.499125</td>
<td>1.499120</td>
<td>1.499120</td>
</tr>
</tbody>
</table>

Table 4. Comparison between the centerline velocities of Parallel-plate channel for different n and Re=500.

<table>
<thead>
<tr>
<th></th>
<th>Present work</th>
<th>Analytical</th>
</tr>
</thead>
<tbody>
<tr>
<td>n=0.50</td>
<td>1.327</td>
<td>1.333</td>
</tr>
<tr>
<td>n=1.00</td>
<td>1.495</td>
<td>1.500</td>
</tr>
<tr>
<td>n=1.25</td>
<td>1.546</td>
<td>1.555</td>
</tr>
</tbody>
</table>

3. Result and discussion

3.1. Non-Newtonian two phase fluid flow in a channel with a cavity

After the performance of computer code was ensured and appropriate grid was selected, different runs were conducted at different states. All calculations for water as Newtonian fluid and solution polymer as non-Newtonian fluid were conducted and constant geometry as Fig.1. Re number varied between 500-1500. The rheological parameter (n) varied between 0.7-1. In following sections, the effects of change in Re number and rheological parameter (n) on the flow parameters were investigated. Coefficients of Carreau-Yasuda model are presented in Tab.5.

Table 5. Coefficients of Carreau-Yasuda model

<table>
<thead>
<tr>
<th>( \rho ) (kg/m(^3))</th>
<th>n</th>
<th>( \lambda ) (s)</th>
<th>( \mu_0 ) (Pa.s)</th>
<th>( \mu_\infty ) (Pa.s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>800</td>
<td>0.7</td>
<td>0.036</td>
<td>0.00345</td>
<td>0.056</td>
</tr>
</tbody>
</table>

3.1.1 The effect of Reynolds number

In this section, we changed Reynolds number only. The geometrical specs were considered constant as shown in Fig.1. Also rheological parameter of non-Newtonian fluid n was considered constant as n=0.8. Fig. 2 indicates streamlines for different Reynolds numbers. Based on the distribution of the streamlines, it can be seen that by increasing Reynolds number, the central vortex moves up and downstream the cavity.

Figure 2. Streamlines for different Re and n=0.8.
As illustrated in Fig. 2 the bottom corners and parts of the near wall regions of the cavity are not covered with the Newtonian fluid. However, by increasing Re, these areas become smaller, but not fully covered. This is due to the rheological nature of non-Newtonian fluids that show resistance to breaking up.

Fig. 3 shows the longitudinal velocity of the Newtonian fluid at \( y = 0 \) from the channel inlet to the channel outlet for various Reynolds numbers. As shown in Fig. 3, by decreasing the Reynolds number, the vertical velocity position moves upward, and the region with negative velocity in the cavity increases. When non-Newtonian fluid enters the channel it mixes with the Newtonian fluid and results in a two-phase flow.

![Figure 3. Velocity distribution at central line of channel for different Re and \( y=0, n=0.8 \).](image)

As we can see in Fig. 4 by increasing Reynolds number initially, the volume fraction of non-Newtonian fluid increases in the channel. Since the flow rate of the Newtonian fluid in channel is constant. As non-Newtonian fluid enters into the channel and decreasing the cross-sectional flow, the velocity of Newtonian fluid increases first, and then, due to the decrease in the flow rate of non-Newtonian fluid into the channel it reaches to steady state.

![Figure 4. Volume fraction of non-Newtonian fluid in the cavity. a) for different Re, n=0.8 and b) Re=1000, n=0.8.](image)

Fig. 5 shows the distribution of non-dimensional longitudinal pressure at the center of the channel. The linear pressure distribution of upstream and downstream cavities represents the flow
developed before and after the cavity. In the cavity area due to the increased cross-section, the velocity decreases and the pressure increases. With regard to the results obtained in the case of pressure, increasing the Reynolds number decreases the pressure flux. The effect of the non-Newtonian fluid is due to the nature of its dilution.

Figure 5. Distribution of longitudinal pressure at the center of the channel for different Re, n=0.8.

3.1.2 The effect of rheological parameter (n)

In this section, we changed rheological parameter (n) only. Geometrical specs were considered constant as shown in Fig.1, also Reynolds number was considered constant as Re=1000. Fig.6 indicates streamlines for different rheological parameter (n). The rheological parameter (n) affects the viscosity and also affects the flow of non-Newtonian fluid and discharging process. Based on the distribution of the streamlines, it can be seen that as n increases, the central vortex moves up and downstream the cavity.

As mentioned earlier, the effect of inertia on fluids with a higher n is higher, and so the fluid is moved upstream easily. In Fig. 6, it can be seen that the elongation rate of the interface area that is caused by fluid displacement in cross-sectional area is higher for fluids with a higher n, and therefore, the mixing of the fluid is better and the discharge time is reduced as we can see in Fig.8. Downstream the cavity two-phase flow occurs and velocity increases, as we can see in Fig.7. The increase in velocity for fluid with higher n is higher.
With the increase in rheological parameter ($n$), the shear rate viscosity decreases and the flowing areas in the dead corners (at bottom of cavity) are enlarged markedly, thus the area with no flow becomes smaller significantly. Because of this reason sweep area and efficiency increase as $n$ increases.

In the cavity area due to increased cross-section, the velocity decreases and the pressure increases. With regard to the results obtained in the case of pressure which are shown in Fig. 9, increasing in $n$, raises the pressure flux. The effect of the non-Newtonian fluid is due to the nature of its dilution.
Figure 9. Distribution of longitudinal pressure at the center of the channel for different n and Re=1000.

4. CONCLUSION

In this paper, non-Newtonian two-phase fluid flow in a channel with cavity was studied. As streamlines show, by increasing Reynolds number the central vortex moves upstream and downstream the cavity. Also by increasing Re, the time of sweeping the non-Newtonian fluid inside the cavity decreases. Also the discharging time of the cavity reduces and the two-phase region downstream the cavity increases. Since the flow rate of the Newtonian fluid in channel is constant, as non-Newtonian fluid enters into the channel and the cross-sectional flow decreases, the velocity of Newtonian fluid increases and the pressure decreases. In the second section, by increasing rheological parameter the central vortex moves upstream the cavity. Also the velocity increases and volume fraction of non-Newtonian fluid after cavity increases. By increasing velocity, pressure decreases.

It is notable that, the changes in the velocity, pressure and volume fraction of fluids inside the channel and cavity are more sensible with changing the Reynolds number instead changing the rheological parameter(n).

Nomenclature

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>D</td>
<td>Deformation tensor, [-]</td>
</tr>
<tr>
<td>f</td>
<td>VOF function, [-]</td>
</tr>
<tr>
<td>H</td>
<td>Channel height, [m]</td>
</tr>
<tr>
<td>n</td>
<td>Rheological parameter, [-]</td>
</tr>
<tr>
<td>P</td>
<td>Pressure, [Pa]</td>
</tr>
<tr>
<td>Re</td>
<td>Reynolds number, [-]</td>
</tr>
<tr>
<td>S</td>
<td>Stress tensor, [-]</td>
</tr>
<tr>
<td>t</td>
<td>Time, [s]</td>
</tr>
<tr>
<td>u,v</td>
<td>Cartesian velocities, [m.s^{-1}]</td>
</tr>
<tr>
<td>u_in</td>
<td>Inlet velocity, [m.s^{-1}]</td>
</tr>
<tr>
<td>X,Y</td>
<td>Cartesian coordinates, [m]</td>
</tr>
</tbody>
</table>

Greek symbols

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ρ</td>
<td>Density, [kg.m^{-3}]</td>
</tr>
<tr>
<td>γ</td>
<td>Shear rate, [s^{-1}]</td>
</tr>
<tr>
<td>μ₀</td>
<td>Zero-shear-rate viscosity, [Pa.s]</td>
</tr>
<tr>
<td>μ∞</td>
<td>Infinite-shear-rate viscosity, [Pa.s]</td>
</tr>
<tr>
<td>λ</td>
<td>Relaxation time, [s]</td>
</tr>
<tr>
<td>τₓₓ, τₓᵧ</td>
<td>Normal and shear stresses, [Pa]</td>
</tr>
</tbody>
</table>

References


