Pressure driven laminar flow of a power-law fluid in a T-channel

O A Dyakova and O Yu Frolov
Tomsk State University, 36 “Lenina” prospect, Tomsk, 634050, Russia
E-mail: dyakova_o@ftf.tsu.ru

Abstract. Planar flow of a non-Newtonian fluid in a T-channel is investigated. The viscosity is determined by the Ostwald–de Waele power law. Motion of the fluid is caused by pressure drop given in boundary sections of the T-channel. On the solid walls, the no slip boundary condition is used. The problem is numerically solved with using a finite difference method based on the SIMPLE procedure. As a result of this study, characteristic flow regimes have been found. Influence of main parameters on the flow pattern has been demonstrated. Criteria dependences describing basic characteristics of the flow under conditions of the present work have been shown.

1. Introduction
A T-channel is a very common element of pipelines networks for transportation of fluids and gases. This element has a variety of engineering applications in irrigation systems, wastewater treatment, biomechanics, pharmaceuticals, as well as, oil and gas, polymer industries, and in many other areas. The fluid flow in a T-channel is characterized by separation of the flow into two parts, and recirculation zones can be formed in the region of the separation. In engineering practice, it is essential to understand the main characteristics of the flow in separating and reattaching flows.

The flow of Newtonian fluids in a T-channel has been studied both experimentally and numerically to obtain basic information of the flow structure and give quantitative estimation. The results of experimental research of a Newtonian fluid flow in a T-channel for various Reynolds number and geometric sizes of the channel are presented in [1-3]. The results of numerical simulation of a turbulence Newtonian flow obtained with using the software package are demonstrated in [4]. Authors of the paper [5] investigated both laminar and turbulence flows of Newtonian and non-Newtonian fluids for various Reynolds number.

With regard to flows of non-Newtonian fluids, steady and unsteady non-Newtonian fluid flows for a range of the Reynolds number Re=10-1000 were investigated in [6]. The data obtained in the study [6] are in close agreement with experimental and numerical results of [1, 7-9]. Authors of [10] solved the problem of the flow of the non-Newtonian fluid describing by the Carreau–Yasuda model in a T-channel for a range of the Reynolds number Re=50-1000. Characteristics of laminar power-law flow in a T-channel for a range of the Reynolds number Re=5-200 and the power-law index $n=0.2-1$ are presented in [11].

In the studies listed above, the flow rate was given in boundary sections of a T-channel. In the research [12], a numerical simulation of the flow of a Newtonian incompressible fluid in channels of complex geometry including fluid flow in a T-channel was performed when the pressure drop was...
given in boundary sections. Also, the authors of [12] proved existence and uniqueness of the solution of such problems.

The objective of the present work is to study of characteristics of the flow of a power-law fluid moving in a T-channel under given pressure drop in boundary sections for a range the Reynolds number Re=5-350 and the power-law index $n=0.6-1.4$.

2. Problem statement and mathematical formulation

Planar steady-state flow of a non-Newtonian incompressible fluid in a T-channel is considered. The flow region is limited by solid walls $MKF, EDC, AB$ (Fig.1). Motion of the fluid is caused by pressure drop given in boundary sections $AM, FE, BC$ of the T-channel. Mathematical problem statement includes momentum and continuity equations which in dimensionless vector form are written as follows:

$$\begin{align}
\text{Re}(U \cdot V) U &= - \text{Re} \nabla p + \nabla \cdot (2\mathbf{E}), \\
\nabla \cdot U &= 0,
\end{align}$$

(1)

where $U$ – velocity vector with components $(U, V)$ in the Cartesian coordinate system, $p$ – the pressure, $\text{Re} = \frac{\rho U_0^2 L}{\mu_0}$ – the Reynolds number, $\rho$ – the fluid density, $\mu_0$ – the power-law consistency index, $\mathbf{E}$ – the rate of deformation tensor. To scale the length and the velocity, $L$ (the width of the boundary section $AM$) and $U_0 = \sqrt{\frac{\Delta p}{\rho}}$ are used, respectively. Here $\Delta p = p_{AM} - p_{BC}$ – the pressure drop between boundary sections $AM$ and $BC$. The dimensionless pressure is defined as $p = \frac{p^* - p_{BC}^*}{\Delta p}$ where the symbol «*» is used to denote dimension values of the pressure.

![Figure 1. Flow region.](image)

The viscosity of non-Newtonian fluid is determined by the Ostwald–de Waele power law [13]:

$$B = (A)^{n-1},$$

where $A = \sqrt{2 \left( \frac{\partial U}{\partial x} \right)^2 + \left( \frac{\partial U}{\partial y} + \frac{\partial V}{\partial x} \right)^2 + 2 \left( \frac{\partial V}{\partial y} \right)^2}$ – dimensionless intensity of the rate of deformation tensor, $n$ – the power-law index.
For pressure, boundary conditions are following:

\[ p_{AM} = 1, \quad x = 0, \quad 0 \leq y \leq 1 \]

\[ p_{BC} = 0, \quad x = L_1 + L_2 + 1, \quad 0 \leq y \leq 1 \]  \hspace{1cm} (3)

\[ p_{FE} = p_1, \quad L_1 \leq x \leq L_1 + 1, \quad y = L_3 \]

On solid walls, no slip boundary condition implying vanishing velocity vector on the solid walls is used:

\[ \mathbf{U} = 0 \]  \hspace{1cm} (4)

The problem solution is reduced to finding both the velocity and pressure fields which satisfy the equations (1), (2) with the given boundary conditions (3), (4).

3. Numerical method

The problem is numerically solved. To find steady-state velocity and pressure fields, the relaxation method is used [14]. This method assumes the addition of time derivative of the function \( \mathbf{U} \) in the equation (1). The obtained system is discretized by the finite difference method based on the SIMPLE procedure [15]; rectangular staggered grid is used.

For pseudo-plastic fluid, there is a singularity of «infinite» viscosity as \( A \rightarrow 0 \). To ensure the stability and accuracy of calculations in the regions of small values of \( A \), additional regularization of the rheological model is necessary. In our study, a modified model of the rheological equation is used. According to this equation, the viscosity is determined by expression \( B = (A + \varepsilon)^{n-1} \). The proposed modification allowing the passage to the power-law model as \( \varepsilon \rightarrow 0 \) extends the range of variation of \( n \) for a stable calculation [16]. The procedure of regularization is justified both physically and from point of view of the computational fluid dynamics. On the one hand, regularization provides a high but finite value of the viscosity \( B \) at low shear rates which corresponds to the physical content of the concept of the viscosity. And on the other hand, the modification of the expression for the viscosity \( B \) facilitates the convergence of the computational algorithm.

To verify the numerical algorithm developed for this study, test calculations were performed. Computations for a sequence of grids were carried out to check the approximate convergence. Calculated value of the velocity in the boundary section \( BC \) was compared with analytical solution of the problem of the steady-state non-Newtonian fluid flow in the planar channel with given pressure drop per length unit \( \bar{\delta} \rho \)

\[ U_0(y) = \text{Re} \left( \delta \rho \left( y - 0.5 \right) \right)^{\frac{n-1}{n}}, \quad 0 \leq y \leq 1 \]  \hspace{1cm} (5)

The value \( \delta \rho \) was numerically determined in vicinity of the boundary \( BC \) where one-dimensional fluid flow is realized. In Tables 1 and 2, dependences of the error \( E \) on the spatial grid step \( h \) and regularization parameter \( \varepsilon \), respectively, are presented. The value \( E \) was calculated by the equation

\[ E = \max_i \left\{ \frac{\left| U_0(y_i) - U_i \right|}{U_0(y_i)} \right\}, \text{ where } U_0(y_i) - \text{the analytic value}, U_i - \text{the calculated value of the velocity} \]

in grid node «\( i \)» on the boundary \( BC \).

Data of Table 1 indicate the approximate convergence. In further calculations, the spatial grid step \( h = 1/40 \) was chosen. It can be seen from Table 2, the error \( E \) decreases with decreasing \( \varepsilon \). However, starting from a certain value of \( \varepsilon \), the error \( E \) increases with decreasing \( \varepsilon \) that indicates a weakening of the role of regularization. Further calculations were carried out for the value \( \varepsilon = 0.01 \).

**Table 1.** Dependence of the error \( E \) on the spatial grid step \( h \) (Re = 40, \( n = 0.6 \)).

| \( h \)  | \( E, \% \) |
|--------|-----|
| 1/10   | 7.8 |
| 1/20   | 3.1 |
| 1/40   | 0.34|
Table 2. Dependence of the error \( E \) on the regularization parameter \( \varepsilon \) (Re = 40, \( n=0.6 \), \( h=1/40 \)).

| \( E \)  | \( E, \% \) |
|--------|-----------|
| 0.1    | 2.2       |
| 0.01   | 0.27      |
| 0.001  | 0.34      |

4. Numerical results

The analysis of the problem statement indicates the characteristics of the flow are depending on geometric sizes of the channel and three parameters: the Reynolds number (Re), the power-law index (\( n \)), and the pressure given on the boundary \( FE \) (\( p_1 \)). In this research, sizes of the T-channel were chosen to avoid the flow influence in vicinity of the corner points \( K \) and \( D \) on the character of the flow near the boundary sections \( AM, FE, BC \) (\( L_1=3 \), \( L_2=4 \), \( L_3=3 \)).

According to numerical calculations, the steady-state flow is formed for the problem investigated. Distribution of the flow characteristics for Re=40 is presented in Fig.2. Close to the boundaries \( AM \), \( FE \), and \( BC \), there is the planar-parallel non-Newtonian flow with parabolic velocity profile (5), while the pressure gradient is constant and directed along the flow direction. Transient flow regions appear in vicinity of the sections with corner points.

![Figure 2](image1.png)

**Figure 2.** Distribution of the flow characteristics for Re=40 (\( n=0.6 \), \( p_1=1.2 \), (a) – the streamline, (b) – the pressure field, (c) –the velocity field \( U \), (d)–the velocity field \( V \)).

Fig. 3 demonstrates influence of the Reynolds number on the steady-state flow pattern, and two characteristic flow regimes can be distinguished for the considered range of variation of this parameter. The first flow regime (\( Re \leq 30 \)): the flow without recirculation zones is realized; the fluid flows into the boundary sections \( AM \) and \( FE \) and flows out into the boundary section \( BC \). Growth of the inertial forces leads to the fluid flow moving from the border \( FE \) begins to press on the flow moving from the border \( AM \) harder. As a result, the recirculation zone in vicinity of the solid wall \( AB \) is formed. Further behavior of the fluid flow is characterized by the growth of recirculation zone sizes with increasing Re. At the critical value of the Reynolds number \( Re=38.45 \) (Fig. 3, c), the recirculation zone closes the cross-section in the direction of the border \( AM \). The second flow regime is realized at \( Re>38.45 \). New flow regime is described by division of the flow moving from the border \( FE \) into two parts. Consequently, the fluid flows into the boundary section \( FE \) and flows out into the boundary...
sections $AM$ and $BC$. The recirculation zone is formed near the border $MN$, and sizes of it decrease with increasing values of the Reynolds number.

**Figure 3.** Streamline distribution for different values of Re ($n=0.6$, $p_1=1.2$, (a) – Re=30, (b) – Re=37, (c) – Re=38.45, (d) – Re=40).

**Figure 4.** Flow rate through the sections $AM$, $FE$ and $BC$ as a function of (a) – Re ($n=0.6$, $p_1=1.2$); (b) – $p_1$ (Re=40, $n=0.6$); (c) – $n$ (Re=40, $p_1=1.2$).
Fig. 4 (a) demonstrates change of the volume flow rate through the sections $AM$, $FE$, and $BC$ with increasing Re. Positive values of the volume flow rate $Q$ correspond to the case when the fluid flows into the T-channel, and negative values of it – when the fluid flows out of the T-channel. It is seen that the absolute volume flow rate through the boundary sections $BC$ and $FE$ increases for the considered range of the Reynolds number. The volume flow rate through the boundary section $AM$ smoothly increase at the beginning, and, after reaching its maximum value, the volume flow rate decreases up to negative values passing through the zero value at Re = 38.45 that corresponds to a change of the flow regime.

Results of research of the head-capacity characteristics depending on the value of the pressure $p_1$ are presented in Fig. 4 (b). Increase of the pressure given on the border $FE$ leads to growth of the absolute value of the flow rate through the boundary sections $BC$ and $FE$. The flow rate through the border $AM$ decreases and reaches zero at the pressure $p_{FE}=1.185$ indicating the second flow regime which describes by separation of the flow moving from the border $FE$. In this case, character of dependence of the flow pattern on the pressure $p_1$ is similar to that observed in the study of influence of the Reynolds number on the flow pattern, as presented above.

Fig. 4 (c) shows change of the flow rate through the boundary sections $AM$, $FE$, and $BC$ with the power-law index $n$. It can be seen that the flow rate curve through the zero value two times that indicates the change of the flow regime.

5. Conclusion

The planar flow of the power-law incompressible fluid in the T-channel has been studied. The motion of the fluid has been caused by pressure drop given in the boundary sections of the T-channel. On the solid walls, the no slip boundary condition has been used.

Investigation of the flow characteristics depending on the Reynolds number (Re=5-350), the pressure given in the border $FE$ ($p_1=1-1.25$), and the power-law index ($n=0.6-1.4$) has been carried out. The range of change of these parameters has been chosen so that the planar-parallel non-Newtonian flow with parabolic velocity profile was realized in vicinity of the boundary sections $AM$, $FE$, and $BC$.

As a result of this study, it has been found that the change of the flow regime occurs with change of these parameters, and the recirculation zone is formed in the stream. Sizes and location of the recirculation zone changes with change of these parameters too. Estimation of the influence of the main parameters on the flow pattern has been performed. Criteria dependences describing characteristics of the flow under conditions of the present work have been presented.

Acknowledgements

This research was supported by the Grant of the President of the Russian Federation for the Young Russian Scientists (MK-710.2017.1).

References

[1] Liepsch D, Moravec S, Rastogi A K and Vlachos N S 1982 Measurements and calculation of laminar flow in a ninety degree bifurcation J. Biomech. 15 473-85
[2] Khodadadi J M, Nguyen T M and Vlachos N S 1986 Laminar forced convective heat transfer in a two-dimensional 90° bifurcation Numer. Heat Transfer 9 677-95
[3] Grace J L and Priest M S 1958 Division of Flow in Open Channel Junctions (Auburn, Ala.: Engineering Experiment Station, Alabama Polytechnic Institute)
[4] Shamloo H and Pirzadeh B 2008 Investigation of characteristics of separation zones in T-Junctions WSEAS Transactions on Mathematics 7 303-12
[5] Benes L, Louda P, Kozel K, Keslerova R and Stigler J 2013 Numerical simulations of flow through channels with T-junction Applied Mathematics and Computation 219 7225-35
[6] Miranda A I P, Oliveira P J and Pinho F T 2008 Steady and unsteady laminar flows of Newtonian and generalized Newtonian fluids in a planar T-junction Int. J. Numerical Methods
in Fluids 57 295-328

[7] Khodadadi J M, Vlachos N S, Liepsch D and Moravec S 1988 LDA measurements and numerical prediction of pulsatile laminar flow in a plane 90-degree bifurcation J. Biomech. Eng. 110 129-36

[8] Khodadadi J M 1991 Wall pressure and shear stress variations in a 90-deg bifurcation during pulsatile laminar flow J. Fluids Eng. 113 111-5

[9] Moshkin N and Yambangwi D 2009 Steady viscous incompressible flow driven by a pressure difference in a planar T-junction channel Int. J. Comput. Fluid Dyn. 23 259-70

[10] Matos H M and Oliveira P J 2013 Steady and unsteady non-Newtonian inelastic flows in a planar T-junction Int. J. Heat and Fluid Flow 39 102-26

[11] Khandelwal V, Dhiman A and Baranyi L 2015 Laminar flow of non-Newtonian shear-thinning fluids in a T-channel Computers & Fluids 108 79-91

[12] Kuznetsov B G, Moshkin N P and Smagulov Sh 1983 Numerical investigation of the viscous incompressible fluid flow in channels of the complex geometry with given pressure (Proceedings of the Institute of the Theoretical and Applied Mechanics of the Siberian Branch of the Academy of Sciences of the USSR Vol. 14) (in Russian)

[13] Shul'man Z P 1975 Convective Heat and Mass Transfer of Rheologically Complex Liquids (Moscow: Énergiya) (in Russian)

[14] Godunov S K and Ryabenkii V S 1987 Difference Schemes (North-Holland: Elsevier Science Ltd)

[15] Patankar S 1980 Numerical Heat Transfer and Fluid Flow (New York: Taylor & Francis)

[16] Shrager G R, Kozlobrodo A N and Yakutenok V A 1999 Simulation of Hydridynamic Processes in the Processing Technology of Polymer Materials (Tomsk: Tomsk State University Publishing) (in Russian)