NUMERICAL CALCULATION USING THE SIMPLE METHOD TO SOLVE THE LAMINAR FLOW PROBLEM IN A SUDDENLY EXPANDED CHANNEL

Shoev M.A1, Rakhmankulov S.A1,
Fergana polytechnic institute,
shoyevmardonbek1@gmail.com
s.raxmonqulov@ferpi.uz

Abstract: In the article, the calculation of flow interruption in the zone of sudden expansion of a two-dimensional channel was carried out. Various flow characteristics were calculated at different Reynolds numbers. Calculations were carried out for laminar flow and based on the numerical solution of the unsteady Navier-Stokes equation. Velocity results at different points of the channel and the length of the piles were studied with the change of Reynolds number after the sudden expansion of the channel, and the velocity and pressure were studied. The relationship between was determined using the SIMPLE method. Various schemes were used to solve this problem numerically. To check the correctness of the results, comparisons with experiments were carried out.
Key words: Navier-Stokes equations, separated flow, laminar flow, SIMPLE method, McCormack scheme, suddenly expanding channel.

1. Introduction

Up to now, modern computers allow studying complex flows by numerically solving more complex problems of hydrodynamics. Such studies are of high importance in the design of various aerodynamic or hydrodynamic devices. The most widely used of such devices in technological processes are diffusers. It is known from observations that flow interruptions may occur in the diffuser. This phenomenon depends on the geometry of the device and several parameters of the flow. Therefore, it is important to know exactly where the flow separation occurs. Flow interruption is of great theoretical and practical importance. Another example of separated flows is a suddenly widened channel flow. As an example of such a flow, we can take the movement of an incompressible viscous fluid in a straight channel with a backward facing step. For the first time, the calculation of stationary two-dimensional laminar separated flows of an incompressible fluid in a straight channel was carried out by

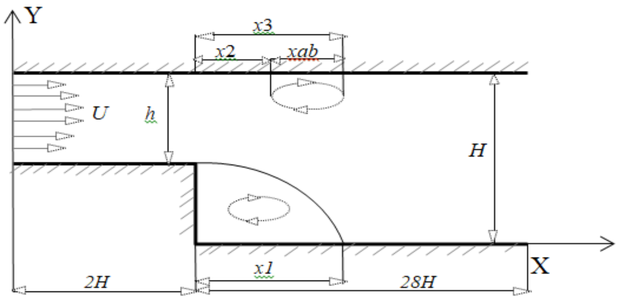
Blasius [1]. He obtained an analytical solution in the form of a series in 1910 Theoretical and experimental results for laminar[2-4] and turbulent motion[5-8] of compressible and incompressible fluid were studied using finite difference schemes for solving Navier-Stokes equations.

Until now, three approaches are mainly used for mathematical modeling of turbulence flows. In the first approach, the turbulent flow is modeled directly using the Navier-Stokes equations. This approach is called Direct Numerical Simulation (DNS) [9,10]. This Navier-Stokes equation is sufficient to describe the turbulent flow. The second numerical method of large-scale modeling is the large-eddy simulation (LES). The LES method is used to calculate flows at significantly higher Reynolds numbers than the DNS method. However, when using the LES[11] method to calculate the near wall, the flows require the use of grids that are similar in their properties to the grids of the DNS method. A third approach to turbulent flow is RANS models, which are closed-loop Reynolds Averaged Navier-Stokes equations. In these models, the system of equations is closed based on various assumptions. Therefore, they are semi-empirical [12]. Currently, there are more than 100 different semi-empirical RANS turbulence models. Based on the results of the analysis, we can conclude that Spalart-Allmaras and Menter SST models have the highest accuracy. These models are widely used because they describe many practical turbulence problems with high accuracy. However, for certain problems, such as flow around bodies of high curvature and flows with strong circulation, these models may give results that do not meet state-of-the-art requirements.

Purpose of work — It consists in comparing the numerical results with the experimental data of the place where the flow separation of the liquid in a straight channel with a sudden one-sided widening channel occurs.

Physical and mathematical formulation of the problem.

Laminar flow in a suddenly widened and flat two-dimensional channel was considered. The physical-mathematical representation of the analyzed laminar flow and its geometry are presented in Figure 1. The smaller channel at the left inlet has a width of h=1m, while the channel at the right outlet has a size H=2h twice the size of the channel inlet. In a smaller channel, a parabolic Poiseuille flow profile was defined for longitudinal velocity-U, transverse velocity-V and pressure-p. Unsteady Navier-Stokes equations for incompressible medium were used to describe the movement of fluid.



1-rasm. When introducing dimensionless quantities, the length scale is assumed to be the width of the large channel -H. The average flow velocity at the inlet is taken as the velocity scale - x1 - the distance from the channel barrier to reattachment of the primary aggregate formed by detachment.Xab - the length of the secondary vortex formed after the step in the upper part of the channel. x2- distance to the head of the secondary vortex x3 the distance to the end of the secondary vortex. The non-stationary Navier-Stokes equations and the continuity equation with constant density in Cartesian coordinates have the following form [14]:

$$\begin{cases} \dfrac{\partial U}{\partial x}+\dfrac{\partial V}{\partial y}=0 \\ \dfrac{\partial U}{\partial t}+U\dfrac{\partial U}{\partial x}+V\dfrac{\partial U}{\partial y}+\dfrac{\partial P}{\partial x}=\dfrac{1}{\mathrm{Re}}\left(\dfrac{\partial^2 U}{\partial x^2}+\dfrac{\partial^2 U}{\partial y^2}\right) \quad \rho=const \\ \dfrac{\partial V}{\partial t}+U\dfrac{\partial V}{\partial x}+V\dfrac{\partial V}{\partial y}+\dfrac{\partial P}{\partial y}=\dfrac{1}{\mathrm{Re}}\left(\dfrac{\partial^2 V}{\partial x^2}+\dfrac{\partial^2 V}{\partial y^2}\right) \end{cases}$$

Here – dimensionless longitudinal and transversely components of the laminar flow velocity vector, respectively, p – dimensionless hydrostatic pressure, Re- Reynolds number.

MacCormack scheme

McCormack's scheme is widely used in computational fluid dynamics. This second-order-accurate finite-difference scheme was developed by Robert W. McCormack in 1969 [15]. This scheme has two stages and looks like thisПредиктор

$$\bar{\Phi}_{i,j}=\Phi^n_{i,j}-\Delta t\left(U^n_{i,j}\frac{\Phi^n_{i+1,j}-\Phi^n_{i,j}}{\Delta x}+V^n_{i,j}\frac{\Phi^n_{i,j+1}-\Phi^n_{i,j}}{\Delta y}\right)+\Delta t\left(\frac{\Phi^n_{i,j+1}-2\Phi^n_{i,j}+\Phi^n_{i,j-1}}{\mathrm{Re}\,\Delta y^2}+\frac{\Phi^n_{i+1,j}-2\Phi^n_{i,j}+\Phi^n_{i-1,j}}{\mathrm{Re}\,\Delta x^2}+\Pi^{\Phi}\right).$$

Corrector

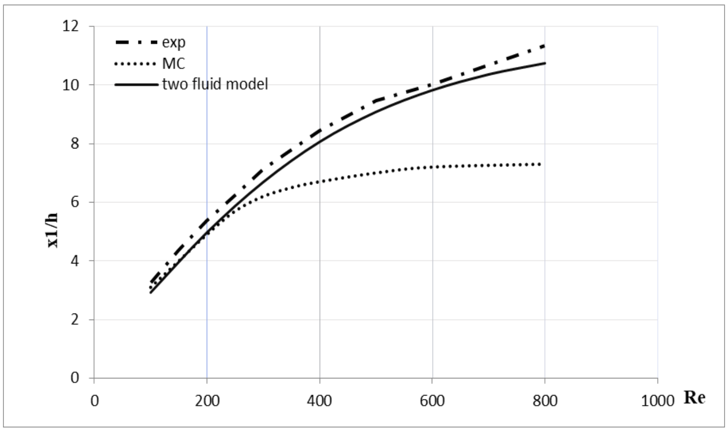
$$\Phi_{i,j}^{n+1}=\frac{1}{2}\left(\bar{\Phi}_{i,j}+\Phi_{i,j}^{n}-\Delta t(\bar{U}_{i,j}^{n}\frac{\bar{\Phi}_{i,j}^{n}-\bar{\Phi}_{i-1,j}^{n}}{\Delta x}+\bar{V}_{i,j}^{n}\frac{\bar{\Phi}_{i,j}^{n}-\bar{\Phi}_{i,j-1}^{n}}{\Delta y})+\Delta t(\frac{\bar{\Phi}_{i,j+1}-2\bar{\Phi}_{i,j}+\bar{\Phi}_{i,j-1}}{\mathrm{Re}\,\Delta y^2}+\frac{\bar{\Phi}_{i+1,j}-2\bar{\Phi}_{i,j}+\bar{\Phi}_{i-1,j}}{\mathrm{Re}\,\Delta x^2}+\Pi^{\Phi})\right).$$

Here $\Phi=\begin{pmatrix}U\\V\end{pmatrix}$, $\Pi^{\Phi}=\begin{pmatrix}\frac{\partial p}{\rho\partial x}\\ \frac{\partial p}{\rho\partial y}\end{pmatrix}$. This scheme has second-order accuracy in time and space, therefore, the approximation error $O\left((\Delta t)^2,(\Delta x)^2,(\Delta y)^2\right)$. It has a stability condition $\left(\frac{U_{\max}\Delta t}{\Delta x}+\frac{V_{\max}\Delta t}{\Delta y}\right)\le 1$.

Initially (predictor) the estimate is found $\overline{\Phi_i^{n+1}}$ magnitude and on $n+1$- m time step, nd then (corrector) the final value is determined Φ_i^{n+1} on $n+1$- m time step. Note that in the predictor it is approximated by forward differences, and in the corrector - by backward differences.

Calculation results and their discussion. $\overline{\Phi_i^{n+1}}$ $n+1$ Φ_i^{n+1} $n+1$

Figure 2 shows graphs comparing numerical results with experimental data using different methods. [16].



Conclusion

The McCormack (MC) scheme was used for the numerical calculation of the one-way widened channel, and the SIMPLE method was used for this scheme. As a result of a detailed numerical study of the flow velocity fields, the formation laws of the swirling flow structure in a flat channel, including secondary vortex formations and unsteady separated flow regimes, are shown. In conclusion, it should be noted that calculation schemes for separated flows can give different results.

References

[1]. Blasius H. Laminare Stromung in Kanalen Wechselnder Breite. Zeitschrift fur Math. und Phys., 1910, vol. 58, no. 10, pp. 225–233.

[2]. Honji H. The starting flow down a step. J. Fluid Mech., 1975, vol. 69, iss. 2, pp. 229–240. DOI: https://doi.org/10.1017/S0022112075001413

[3]. Синха С.П., Гупта А.К., Оберай М.М. Ламинарное отрывное обтекание уступов и каверн. Ч. 1. Течение за уступом. Ракетная техника и космонавтика, 1981, т. 19, № 12, с. 33–37.

[4]. Armaly B.F., Durst F., Pereira J.C.F., et al. Experimental and theoretical investigation
of backward-facing step flow. J. Fluid Mech., 1983, vol. 127, pp. 473–496. DOI: https://doi.org/10.1017/S0022112083002839

[5]. Чжен П. Отрывные течения, М., Мир, 1972.

[6]. Гогиш Л.В., Степанов Г.Ю. Турбулентные отрывные течения. М., Наука, 1979.

[7]. Le H., Moin P., Kim J. Direct numerical simulation of turbulent flow over a backward-facing step. J. Fluid Mech., 1997, vol. 330, pp. 349–374. DOI: https://doi.org/10.1017/S0022112096003941

[8]. Durst F., Melling A., Whitelow J.H. Low Reynolds number flow over a plane symmetric sudden expansion. J. Fluid Mech., 1974, vol. 64, iss. 1, pp. 111–118. DOI: https://doi.org/10.1017/S0022112074002035

[9] Versteegh T A, Nieuwstadt F T 1997 Turbulent Budgets of Natural Convection in an Infinite,Differentially Heated, Vertical Channel Intern. J. Heat Fluid Flow. vol 19 P 135

[10] Boudjemadi R, Maupu V, Laurence D, Le Quere P 1996 Direct Numerical Simulation of Natural Convection in a Vertical Channel: A Tool for Second-Moment Closure Modelling Proc. Engineering Turbulence Modelling and Experiments 3 (Amsterdam: Elsevier) P 39

[11] Peng S-H, Davidson L 2001 Large Eddy Simulation of Turbulent Buoyant Flow in a Confined
Cavity Intern. J. Heat Fluid Flow. vol 22 P 323

[12] Cristopher R Responsible NASA official. “Turbulence modeling Resource. NASA Langley
Research Center”, http://turbmodels.larc.nasa.gov. (data obrashcheniya 04.04.2019)

[13]. Acrivos A., Schrader M.L. Steady flow in a sudden expansion at high Reynolds numbers. Phys. Fluids, 1982, vol. 25, iss. 6, pp. 923–930. DOI: https://doi.org/10.1063/1.863844

[14]. Лойцянский Л.Г. Механика жидкости и газа // Масква. Наука, 1987.-678 с.

[15]. MacCormack R. W. (1969). The Effect of Viscosity in Hypervelocity Impact Cratering. — AIAA Paper 69—354, Cincinnati, Ohie.

[16]. Маликов З.М., Мадалиев М.Э. Численное моделирование течения в плоском внезапно расширяющемся канале на основе новой двужидкостной модели турбулентности. Вестник МГТУ им. Н.Э. Баумана. Сер. Естественные науки, 2021, № 4 (97), с. 24–39. DOI: https://doi.org/10.18698/1812-3368-2021-4-24-39