Volume 4, Issue 6, June - 2026 **ISSN(E):** 2938-3773

IMPLEMENTATION OF A TWO-FLUID TURBULENCE MODEL IN THE COMSOL MULTIPHYSICS SOFTWARE PACKAGE

S. I. Khaydarov
Assistant, AndSTI

Abstract
This article examines the feasibility of implementing a new two-fluid turbulence model in the COMSOL Multiphysics software package. This program is a powerful mathematical tool for solving various CFD engineering problems. One distinguishing feature of this program compared to other similar CFD packages is the ability to implement a new model by writing differential equations directly into mathematics interface. To validate the model, a problem of turbulent flow past a flat plate is considered, and the results are compared with known experimental data presented by NACA.

Keywords: Turbulent flow, flat plate, two fluid turbulence model, COMSOL Multiphysics, Reynolds number, friction coefficient.

Introduction

Currently, high-performance computers allow engineers to study complex flow flows by numerically solving hydrodynamic equations using one of the existing computational methods. Such studies are of great importance for the study of aero - or hydrodynamic processes, or for the design of various devices. It is known that in the vast majority of cases, the flows are turbulent in nature. Turbulence is also of great importance for the study of nano-fluids. Therefore, to study such flows, a reliable mathematical model of turbulence should be used as a tool. There are more than 120 different models in the world. However, each model is able to adequately describe only a certain class of turbulent flows. Therefore, there is still no universal model of turbulence and the creation of a reliable mathematical model of turbulence is an important problem of hydrodynamics.

Currently, there are several approaches for constructing mathematical models of turbulence. The first approach is the hypothesis about the applicability of the Navier-Stokes equations to describe turbulence. All models built on the basis of this approach are called direct modeling methods (DNS) and large vortices (LES). However, to implement these methods, it is necessary to solve non-stationary equations with calculation cells of less than Kolmogorov scale in a three-dimensional formulation and it is necessary to perform integration in very small time steps. Therefore, the use of powerful supercomputers is required and their wide practical application, according to experts, can begin only at the end of this century.

The two-fluid turbulence model has recently gained increasing popularity [1, 2]. This turbulence model is based on the dynamics of two fluids, which, unlike the Reynolds approach, leads to a closed system of equations. A unique feature of this model is its ability to describe complex anisotropic turbulent flows.

Web of Discoveries: Journal of Analysis and Inventions
webofjournals.com/index.php/3

The problem under consideration is of great importance for aviation and rocket and space technology. In [1], flow past a flat plate was studied using the two-fluid model. In this case, a simplified, parabolic system of equations was used, i.e. the pressure was assumed to be constant and diffusion terms in the longitudinal direction were neglected. However, the pressure cannot be considered constant in all streamlined flows. For example, in many technical devices, the flow can occur in confined spaces. In the work [3], a two-fluid model for this problem was used, a complete system of turbulence equations, and the results were compared with experimental data. A description of this problem is presented in the NASA database [4]. Currently, modern software packages Ansys, Solidworks and Comsol Multiphysics are used to simulate hydro and aerodynamic processes. Using these programs, it is possible to solve many problems of hydro and aerodynamics. Therefore, in this work, a two-fluid model is introduced into the Comsol Multiphysics code and a solution is obtained for the problem of flow past a flat plate.

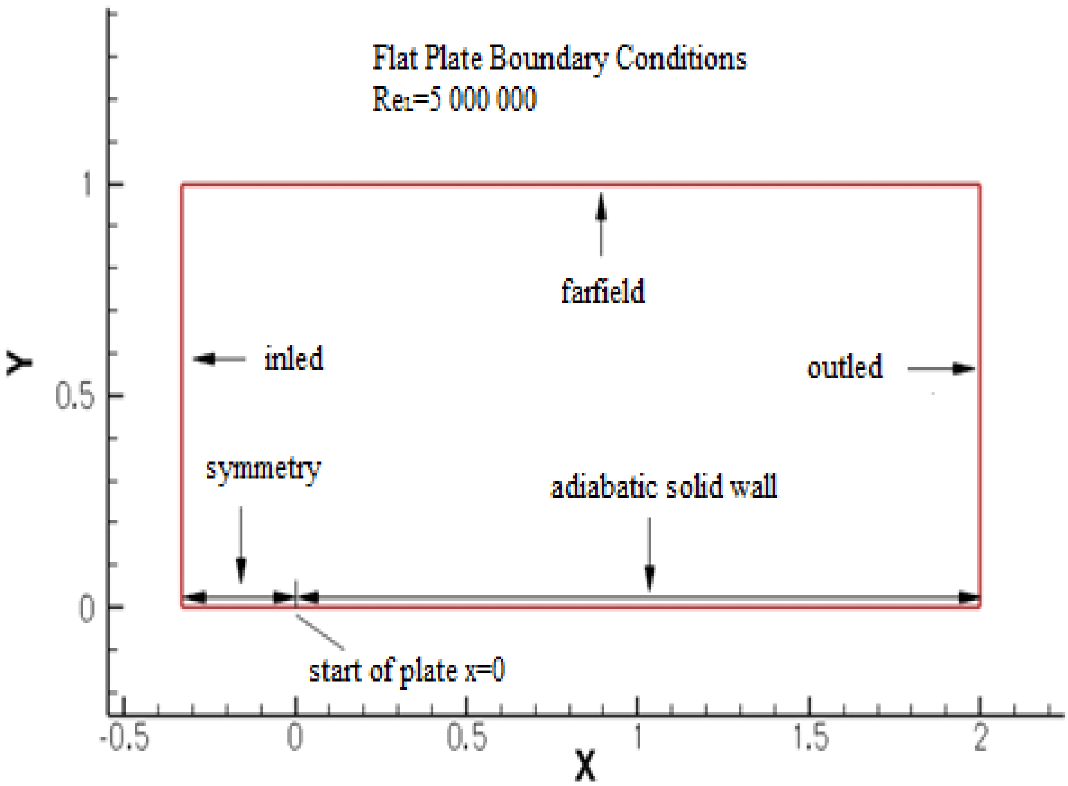

Figure. 1. Schematic diagram of the computational domain in a flat channel

A two-dimensional turbulent flow in a flat channel is considered. The physical picture of the analyzed flow and the configuration of the computational domain are shown in Figure 1. The unsteady system of equations for the turbulent two-fluid model has the following form [1]:

$$
\begin{cases}
\dfrac{\partial \rho}{\partial \tau}+\dfrac{\partial \rho V_j}{\partial x_j}=0,\\[2mm]
\dfrac{\partial \rho V_i}{\partial \tau}+\dfrac{\partial \rho V_j V_i}{\partial x_j}+\dfrac{\partial p}{\partial x_i}=\dfrac{\partial P_{ji}}{\partial x_j}-\dfrac{\partial \rho \vartheta_j \vartheta_i}{\partial x_j},\\[2mm]
\dfrac{\partial \rho \vartheta_i}{\partial \tau}+\dfrac{\partial \rho V_j \vartheta_i}{\partial x_j}=-\rho\vartheta_j\dfrac{\partial \rho V_j}{\partial x_j}+\dfrac{\partial}{\partial x_j}\left[\rho\nu_{ij}\left(\dfrac{\partial \vartheta_i}{\partial x_j}+\dfrac{\partial \vartheta_j}{\partial x_i}\right)\right]+\rho C_s\left[rot\vec{V}\times\vec{\vartheta}\right]_i-\rho K_f\vartheta_i,\\[2mm]
\dfrac{\partial \rho T}{\partial \tau}+\dfrac{\partial \rho V_j T}{\partial x_j}=\dfrac{\partial \rho}{\partial \tau}+V_j\dfrac{\partial p}{\partial x_j}+\dfrac{\partial}{\partial x_j}\left(\rho k\dfrac{\partial T}{\partial x_j}\right)+\dfrac{\mu}{2}\left(S_{ij}S_{ij}+s_{ij}s_{ij}\right)-\dfrac{2\mu}{3}\left[\left(\dfrac{\partial V_i}{\partial x_i}\right)^2+\left(\dfrac{\partial \vartheta_i}{\partial x_i}\right)^2\right]-\dfrac{\partial \rho\vartheta_j t}{\partial x_j},\\[2mm]
\dfrac{\partial \rho t}{\partial \tau}+\dfrac{\partial \rho V_j t}{\partial x_j}=\vartheta_j\dfrac{\partial p}{\partial x_j}-\rho\vartheta_j\dfrac{\partial T}{\partial x_j}+\dfrac{\partial}{\partial x_j}\left(\rho k_j\dfrac{\partial t}{\partial x_j}\right)+\mu S_{ij}s_{ij}-\dfrac{4\mu}{3}\dfrac{\partial V_i}{\partial x_i}\dfrac{\partial \vartheta_i}{\partial x_i}-\rho K_t t.
\end{cases}
\qquad (1)
$$

$$
\nu_{ij}=3\nu+2\left|\frac{\vartheta_i\vartheta_j}{\operatorname{def}(V)}\right|\ for\, i\neq j,\quad \nu_{ii}=3\nu+\sum_{k=1}^{3}\frac{2I_k}{I}\left|\frac{\vartheta_k\vartheta_k}{\operatorname{def}(V)}\right|,\quad k_j=3k+2\left|\frac{t\vartheta_j}{\nabla T}\right|,
$$

$$
I_k=\frac{\partial\rho\vartheta_k}{x_k},\quad I=|I_1|+|I_2|+|I_3|,\ S_{ij}=\frac{\partial\vec{V}_i}{\partial x_j}+\frac{\partial\vec{V}_j}{\partial x_i},\ s_{ij}=\frac{\partial\vartheta_i}{\partial x_j}+\frac{\partial\vartheta_j}{\partial x_i},\ P_{ij}=\mu\left(\frac{\partial V_i}{\partial x_j}+\frac{\partial V_j}{\partial x_i}\right)-\frac{2}{3}\delta_{ij}div\vec{V}.
$$

The method for deriving this system, as well as the model constants, are presented in [1–2].
In this study, mesh refinement was used near the plate wall in the vertical direction and in the longitudinal direction at the plate origin. The computational mesh is shown in [4].
For numerical implementation, system of equations (1) was reduced to dimensionless form by relating all velocities to the incoming flow velocity, and all linear dimensions to the plate length L. Obvious no-slip boundary conditions were imposed at the wall. At the outlet, extrapolation conditions were adopted for the horizontal, vertical, and relative velocities. At the inlet, uniform profiles of the longitudinal velocity component were applied with $V_x=U_0$, while the transverse velocity component and pressure were equal to zero, $V_y= P= 0$.
For the numerical implementation of the system of equations (1), the following conditions were set at the input for the relative velocities (disturbance): $\upsilon_x=0.03$, $\upsilon_y=0$.
Below are shown comparisons of the obtained numerical results with known experimental data. Figure 2. shows: a) the dependence of the friction coefficient on the change in the plate distance, b) the dimensionless longitudinal flow velocity depending on the dimensionless distance to the plate, as well as the experimental results [4].

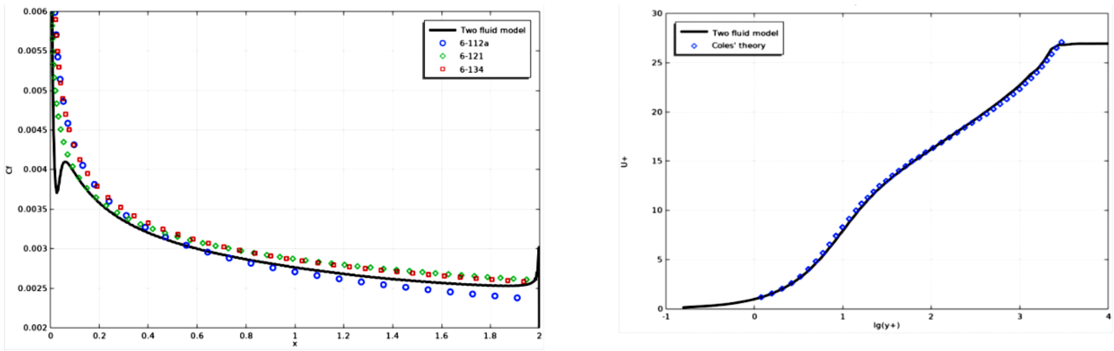

Figure 2. Dependence of the friction coefficient along the plate a), profile of the dimensionless longitudinal flow velocity on the dimensionless distance to the plate b).

Web of Discoveries: Journal of Analysis and Inventions
webofjournals.com/index.php/3

Web of Discoveries: Journal of Analysis and Inventions
webofjournals.com/index.php/3

Figure 3. shows the profiles of dimensionless longitudinal velocity at different Reynolds numbers in two sections: a) x=0.97 m and b) x=1.97 m.

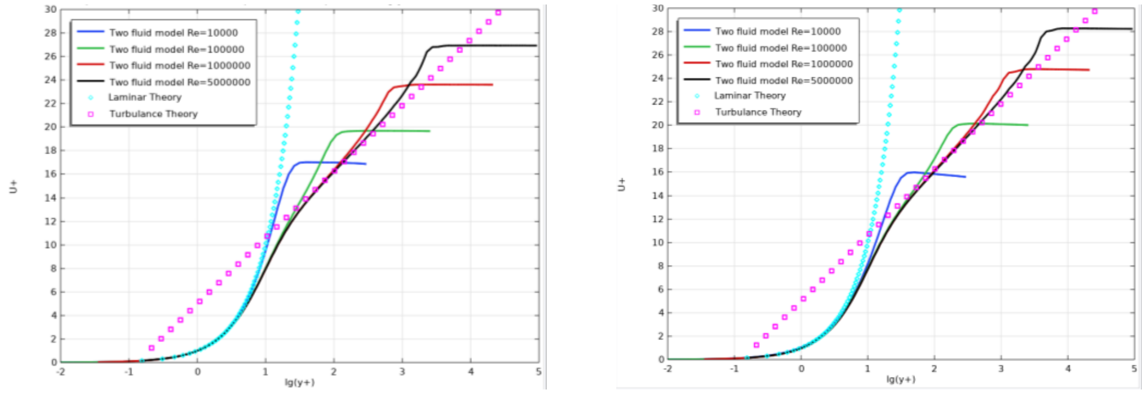

Figure 3. Dimensionless longitudinal flow velocity at different Reynolds numbers.

The figures show that the model describes both laminar and turbulent zones well at all Reynolds numbers.

This paper presents numerical solutions for incompressible viscous fluid flow past a flat plate using a two-fluid turbulence model in the Comsol Multiphysics software package. The results of the two-fluid turbulence model are shown to be in good agreement with experimental data.

References

1. Malikov Z. "Mathematical Model of Turbulence Based on the Dynamics of Two Fluids". Applied Mathematical Modeling. V.82, P. 409-436. 2020

2. Malikov Z.M. Mathematical model of turbulent heat transfer based on the dynamics of two fluids. Applied Mathematic Modeling. 91 2021 186–213. https://orcid.org/0000-0001-9038-5407 3.

3. Madaliev, M., Yunusaliev, E., Usmanov, A., Usmonova, N., & Muxammadyoqubov, K. (2023). Numerical study of flow around flat plate using higher-order accuracy scheme. In E3S Web of Conferences (Vol. 365, p. 01011). EDP Sciences.

4. Cristopher R. Responsible NASA official. "Turbulence modeling Resource. NASA Langley Research Center", http://turbmodels.larc.nasa.gov.