Journal of Computational and Applied Mathematics 270 (2014) 451–461

Contents lists available at ScienceDirect

Journal of Computational and Applied Mathematics

journal homepage: www.elsevier.com/locate/cam

On numerical simulation of three-dimensional flow problems by finite element and finite volume techniques

Petr Sváček [a,*], Petr Louda [b], Karel Kozel [a]

[a] Czech Technical University in Prague, Faculty of Mechanical Engineering, Department of Technical Mathematics, Karlovo nám. 13, 121 35 Praha 2, Czech Republic

[b] Institute of Thermomechanics, Czech Academy of Sciences, Dolejškova 5, 182 00 Praha 8, Czech Republic

ARTICLE INFO

Article history:
Received 2 October 2013
Received in revised form 4 December 2013

Keywords:
Backward facing step
EARSM turbulence model
Finite element method
Finite volume method

ABSTRACT

This paper is interested in the numerical approximation of the turbulent 3D incompressible flow. The turbulent flow is mathematically modeled using the Reynolds averaged Navier–Stokes (RANS) equations and two classes of the turbulent models are considered. RANS equations are approximated by two numerical techniques, the finite volume and the finite element methods.

The finite element approximation on general 3D domains using general meshes consisting of hexahedrons as well as tetrahedrons, pyramids and prisms is described. The definition of the continuous piecewise trilinear/linear finite element space is given, and the stabilization based on the streamline-upwind/Petrov–Galerkin method together with the pressure stabilizing/Petrov–Galerkin techniques is used. The turbulence k–ω model is approximated on the finite element spaces, and the nonlinear stabilization technique is applied. Furthermore, the finite volume technique is used for the approximation of the RANS equations. The turbulent k–ω or the explicit algebraic Reynolds stress models are used. The numerical solution is carried out by the implicit finite volume method. The artificial compressibility method is used to solve the incompressibility constraint. The numerical results are shown.

0. Introduction

Mathematical modeling is important in many engineering problems of fluid and solid mechanics. The use of proper mathematical model is usually crucial to get the correct results. Recently, also the numerical simulations are being in the practice for the solution of a large number of technical problems. Due to the increase of the computer power, the simulations of complex mathematical models is possible, cf. [1,2]. Particularly, the turbulent flow under different flow conditions is modeled and numerically approximated in various applications, see e.g. [3,4]. The effect of the numerical errors and/or the applied mathematical model is usually not investigated. Here, the approximation of the turbulent flow in a three dimensional channel is considered, with possible modifications of the geometrical configuration. The step shape is either perpendicular or inclined, and the top wall inclination angle can vary from being parallel to the bottom wall.

For such a geometrical configuration a large number of the experimental as well as the numerical results are available. The measurement of both the laminar and the turbulent flows over a perpendicular backward facing step was published in [5]. The step was relatively confined, the expansion ratio (ER) of the step was approximately $ER \approx 2$. Driver and Seegmiller considered less confined flow over a perpendicular step with the expansion ratio $ER = 9/8$, see [6], where the deflection

* Corresponding author. Tel.: +420 224357413.
E-mail addresses: psvacek@gmail.com, Petr.Svacek@fs.cvut.cz (P. Sváček).

0377-0427/$ – see front matter © 2013 Elsevier B.V. All rights reserved.
http://dx.doi.org/10.1016/j.cam.2013.12.019

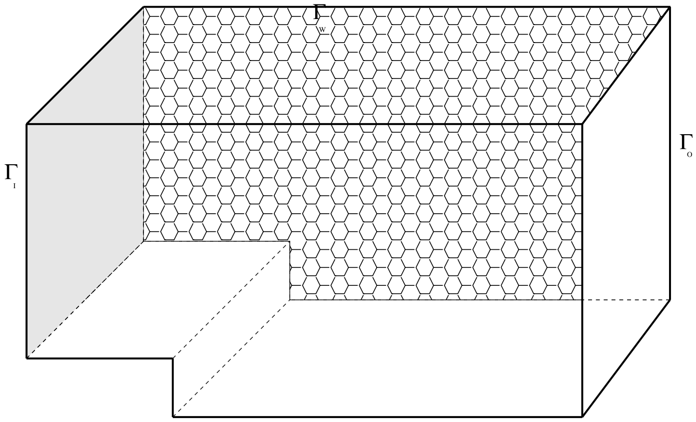

Fig. 1. An example of 3D computational domain Ω with the inlet (Γ_I), the outlet (Γ_O), and the wall (Γ_W) parts of the boundary.

of the upper wall of the channel was considered, superimposing pressure gradients to the flow. Another generalization of the geometry was considered in [7], where the measurement of the flow over steps with an inclined wall was presented. Recently, the PIV measurement of the flow over the perpendicular step in a narrow channel was published in [8].

For the mathematical modeling of the turbulent flow see, e.g., [9–11]. Except the direct numerical simulations or the large eddy simulations, the mathematical models are usually based on the Reynolds averaged Navier–Stokes (RANS) equations containing the so-called Reynolds stresses. In order to closure the system, these stresses are modeled using a so-called turbulence model. Typically, the eddy viscosity approximation of the Reynolds stresses are used in CFD, cf. [12–14]. The other possibility is to use some Reynolds stress model, here we consider the explicit algebraic Reynolds stress (EARSM) constitutive relation, where for the turbulent scales the k–ω model is used, cf. [15].

The governing equations are then approximated by the in-house implementation of the finite-element and the finite volume methods. The numerical results for some of the above cases can be approximated also as 2D flow near the center-plane. The two-dimensional numerical approximation is possible, see also authors' previous works, e.g. [16,17]. Nevertheless, for the turbulent flow approximation the data from 2D simulations indicates that the eddy viscosity turbulence models do not predict consistent 2D and 3D results, [18]. In this situation, the use of explicit algebraic stress tensor model(EARSM) is the option, as it captures correctly the corner vortices as well as the length of the separation for both the 2D and the 3D central plane.

The paper is structured as follows: after the introduction, the mathematical model consisting of Reynolds averaged Navier–Stokes equations and the two classes of turbulence models is described. Furthermore, the problem of the finite element approximation on general 3D triangulation is considered, the stabilized formulation is defined and the solution algorithm of the nonlinear problem is given. Last, the finite volume method is briefly presented and the numerical results are shown.

1. Mathematical model

For the main concepts of the modeling of the turbulent flow see, e.g., [14,11,9]. In this paper we consider the incompressible viscous turbulent flow in the computational domain $\Omega \subset \mathbb{R}^3$ (see Fig. 1) governed by the Reynolds averaged Navier–Stokes (RANS) equations written in the form

$$\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u}\cdot\nabla)\mathbf{u} + \nabla p = \nabla\cdot\left(2\nu\mathbf{S} - \boldsymbol{\mathcal{T}}\right), \quad \nabla\cdot\mathbf{u} = 0, \tag{1}$$

where $\mathbf{u} = \mathbf{u}(x,t)$ denotes the mean part of the velocity vector, $\mathbf{u} = (u_1, u_2, u_3)$, $p = p(x,t)$ denotes the mean part of the kinematic pressure, ν is the constant kinematic viscosity (i.e. the viscosity divided by the constant fluid density ρ), t denotes time, $\mathbf{S} = \frac{1}{2}(\nabla\mathbf{u} + \nabla^T\mathbf{u})$ is the mean rate of the strain tensor with the components

$$S_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right),$$

and $\boldsymbol{\mathcal{T}} = (\tau_{ij})$ is the Reynolds stress tensor, cf. [14].

System of equations (1) is equipped with the initial

$$\mathbf{u}(x,0) = \mathbf{u}^0(x) \quad x \in \Omega, \tag{2}$$