Engineering Science and Technology, an International Journal 19 (2016) 904–910

HOSTED BY

Contents lists available at ScienceDirect

Engineering Science and Technology, an International Journal

journal homepage: http://www.elsevier.com/locate/jestch

Full Length Article

Numerical study on flow separation in 90° pipe bend under high Reynolds number by k-ε modelling

Prasun Dutta *, Sumit Kumar Saha, Nityananda Nandi, Nairit Pal

Department of Aerospace Engineering and Applied Mechanics, Indian Institute of Engineering Science and Technology, Shibpur, Howrah 711103, India

ARTICLE INFO

Article history:
Received 7 October 2015
Received in revised form
15 December 2015
Accepted 15 December 2015
Available online 25 January 2016

Keywords:
90° pipe bend
k-ε turbulence model
Turbulent flow
Flow separation

ABSTRACT

The present paper makes an effort to find the flow separation characteristics under high Reynolds number in pipe bends. Single phase turbulent flow through pipe bends is investigated using k-ε turbulence model. After the validation of present model against existing experimental results, a detailed study has been performed to study the influence of Reynolds number on flow separation and reattachment. The separation region and the velocity field of the primary and the secondary flows in different sections have been illustrated. Numerical results show that flow separation can be clearly visualized for bend with low curvature ratio. Distributions of the velocity vector show the secondary motion clearly induced by the movement of fluid from inner to outer wall of the bend leading to flow separation. This paper provides numerical results to understand the flow characteristics of fluid flow in 90° bend pipe.

1. Introduction

Pipe bends are the most important part of any pipeline network system as these provide flexibility in routing. Investigations of the flow through bends are of great significance in understanding and improving their performance and minimizing the losses. It is already well known that the flow of incompressible viscous fluids through pipe bends is characterized by flow separation, secondary flow and unsteadiness, which are dependent on Reynolds number as well as the radius of curvature of the bend. Whenever a fluid flows through a bend, there is a radial pressure gradient developed by the centrifugal force acting on the fluid. Because of this, a double spiral flow field and a pair of counter-rotating vortices can also be observed inside the bend i.e. because of the presence of pressure gradient, fluid at the centre of pipe moves towards the outer side and comes back along the wall towards the inner side. Now if the bend curvature ratio is very small ($Rc/D \leq 1.5$), the adverse pressure gradient near the inner wall and immediately downstream of the bend may lead to flow separation, giving rise to a large increase in pressure losses [1–3]. However, the flow characteristics of incompressible flows in pipe bends are not fully clarified yet. Accurate estimation of mass flow rate and losses is critical for most incompressible flow systems. The applications of water-flows through pipe bends are found in many engineering applications. Some of the excellent reviews bear testimony to this fact. A number of researchers have investigated turbulent flows in pipe bends by means of theoretical, experimental and numerical methods [4–9]. To perform numerical simulation of fluid flow in curved pipes, on the other hand, the Navier–Stokes equation has to be expressed in curvilinear or body fitted coordinate system. A very useful database for direct numerical simulation (DNS) and large eddy simulation (LES) on pipe bend is provided by two studies [10,11]. Recently, in the nuclear sector due to the fatigue by the unsteady motion of the vortices, this has also attracted the interest of the researchers [2,12,13]. Hence, it is interesting to see the flow separation and reattachment under high Reynolds number. Micro and nano size particle erosion in 90° pipe bend and over the backward-facing steps were studied numerically [14–18]. Very recently, studies on turbulent mixed convection heat transfer [19–25] attracted the interest of investigators; many researchers used Lattice Boltzmann methods to solve natural convection heat transfer problem [26,27]. Magneto hydrodynamic flow (MHD) has attracted much interest of researchers in recent years due to the effect of magnetic field on the boundary layer flow control [28–33]. Different exact and approximate techniques have also been used to solve the different problems in fluid mechanics [34–38]. In this paper, the flow separation in the most common 90° pipe bend is studied by numerical methods based on computational fluid dynamics. The paper is structured in the following fashion. Section 1 gives a brief idea on the previous research works and motivation for the present work. Section 2 contains the necessary theoretical background. Problem definition with validation is provided in section 3. Section 4 contains the study on various parameters affecting the flow pattern and is followed by summary bulletin of the study under section 5.

* Corresponding author. Tel.: +91 33 2668–4561 to 63 (Extn.: 277).
E-mail address: pd.iiest@gmail.com (P. Dutta).
Peer review under responsibility of Karabuk University.

http://dx.doi.org/10.1016/j.jestch.2015.12.005
2215-0986/

2. Governing equations and numerical methodology

Three dimensional Reynolds Averaged Navier–Stokes (RANS) equations are solved using the segregated implicit solver. The right choice of a turbulence model is critical when an industrial turbulent flow problem is faced, especially when this problem involves three dimensional flow phenomena, which needs an accurate modelling. The second order scheme is used for the U-RANS equations calculations, with a pressure velocity coupling achieved using SIMPLE algorithm. The time step size used in the present study is 0.001 s with 1000 time steps. The default under relaxation factors were used to aid convergence for all models.

The governing equations for incompressible fluid flow with constant properties are

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = f_i - \frac{1}{\rho}\frac{\partial p}{\partial x_i} + \upsilon \frac{\partial^2 u_i}{\partial x_j \partial x_j} \tag{2}$$

Equations (1) and (2) are conservations of mass and momentum, respectively; f_i is a vector representing external forces, υ is the kinematic viscosity.

2.1. Turbulence model

It is well known that turbulent flows are basically designated by the fluctuations of the velocity fields. Different transported quantities such as momentum, energy, etc. also fluctuate for this fluctuation of velocity field and these fluctuations can be of very high frequency and small scale; they are very difficult and computationally crucial to analyse directly in industrial engineering calculations. The turbulence model needs to be selected based on some considerations, e.g., the physics of the flow, the insight into the capabilities and limitations of turbulence models, the attempt for the specific problem by other researchers, the accuracy needed, the available computational resources, and time.

The k-ε turbulence model is adopted for the present study as k-ε turbulence model performs better for both single-phase and two-phase flows in pipe bend [3,39–43]. In this model, the turbulence kinetic energy (k) and the turbulence dissipation rate (ε) are solved to determine the coefficient of turbulent viscosity (μ_t).

Transport equation for k-epsilon

$$\frac{\partial(pk)}{\partial t} + \frac{\partial(pku_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\frac{\mu_t}{\sigma_k}\frac{\partial k}{\partial x_j}\right] + 2\mu_t E_{ij} E_{ij} - \rho\varepsilon \tag{3}$$

$$\frac{\partial(p\varepsilon)}{\partial t} + \frac{\partial(p\varepsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\frac{\mu_t}{\sigma_\varepsilon}\frac{\partial k}{\partial x_j}\right] + C_{1\varepsilon}\frac{\varepsilon}{k} 2\mu_t E_{ij} E_{ij} - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \tag{4}$$

where u_i represents velocity component in corresponding direction, E_{ij} represents component of rate of deformation, and μ_t represents eddy viscosity.

Equations (3) and (4) also consist of some adjustable constants [44]; these are as follows

$$C_\mu = 0.09 \quad \sigma_\kappa = 1.00 \quad \sigma_\kappa = 1.00 \quad \sigma_\varepsilon = 1.30 \quad C_{1\varepsilon} = 1.44 \quad C_{2\varepsilon} = 1.92$$

3. Problem definition

The problem that is considered here is the fluid flow through 90° pipe bends having inner diameter of 0.01 m with curvature ratio (Rc/D) = 1 for different Reynolds numbers ranging from 1×10^5 to 10×10^5. The inlet length of straight pipe in the calculations was set up 50D for all cases to save computational time. The fluid medium was air having density (ρ) of 1.2647 kg/m^3 and dynamic viscosity (μ) of 1.983×10^{-5} kg/m-s for validation purpose and water having density (ρ) of 990.2 kg/m^3 and dynamic viscosity (μ) of 0.0006 kg/m-s for the present study with working temperature of 300 K in both cases. Three dimensional structured mesh was used containing hexahedron elements, which was optimized via a grid-independence study. The bend geometry and mesh are shown in Fig. 1a and b respectively. It is defined that the axial direction downstream the bend is x-coordinate, the direction from inner core to outer core of the bend is y-coordinate and the perpendicular direction to x and y is z-coordinate.

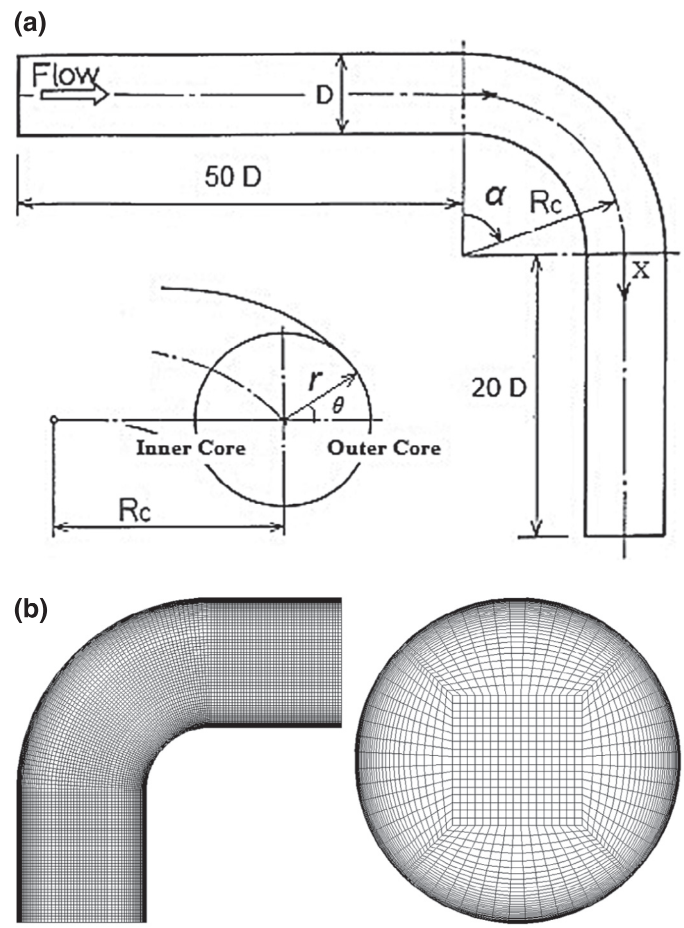

Fig. 1. Schematic diagram of the bend geometry and present model with computational grid.

3.1. Validation

At the very beginning of our study, our model and simulation setup are first validated against the existing experimental and numerical data in References [3,7,45]. For that intension, same geometrical configuration is adopted. In their experiment, the authors of the previously mentioned studies used a circular cross sectioned 90° bend with a curvature ratio (R_c/D) of 2 and the measurements of velocities were performed at a Reynolds number of 6×10^4. For the validation of our present model, the simulation is performed on a computational mesh containing total 2.85 million hexahedron elements, which was optimized via a grid-independence study, see Fig. 1a. The value of non-dimensional distance from wall (Y^+) is strictly controlled using standard wall treatment function ($30 < Y^+ < 90$ for a near wall cell used for present study). The mean