International Journal of Enhanced Research in Science Technology & Engineering, ISSN: 2319-7463
Vol. 3 Issue 7, July-2014, pp: (124-132), Impact Factor: 1.252, Available online at: www.erpublications.com

CFD Analysis of Flow Structure of the Dynamic Turbulent Flow Field through a Channel Provided With Baffles

Sitender Dahiya[1], Sunil Dhingra[2], Manoj Kumar[3]
[1,3]Student, [2]Assistant Professor
[1,2,3]Department of Mechanical Engineering, UIET Kurukshetra University, Kurukshetra, India

---

Abstract: A study of flow structure of airflow through a rectangular channel in the presence of baffles was conducted using FLUENT. The main objective of this work is to examine the dynamic behavior of a turbulent air flow passing through a rectangular channel, containing baffles attached to the two walls of a channel. Baffles were arranged at different positions. Comparison of flow structure of plane channel, channel with two baffles and with four baffles is conducted in this article. The governing equations based on k - ε model at low Reynolds number used velocity profiles were obtained for all the geometry considered and selected for different sections, namely, upstream, downstream and between the baffles and the skin friction coefficients were obtained for different sections.

Keywords: Turbulent flow, plane channel, rectangular baffles, Finite volume method simple algorithm.

---

1. Introduction

Turbulent flow in complex geometries receives considerable attention due to its Importance in many engineering applications and has been the subject of interest for many researchers. Some of these include the energy conversion systems found in same design of nuclear reactor, heat exchangers, solar collectors and cooling of industrial machines and electronic components. Flow in channels with baffle plates occurs in many industrial applications such as heat exchangers, chemical reactors, filtration and desalination. These baffles increase the turbulence. Many work have been done in recent years, on investigations of heat transfer process and flow structure in a channel in order to improve the accuracy of prediction of performance of heat exchanger. Some works are in the improvement of flows around baffles. Some studies are done experimentally and some are numerically techniques. Extensive studies of turbulent flow and heat transfer using rectangular:-

Dr. Qasim S. Mehdi investigates the turbulent flow and heat transfer through a duct with baffle plates. The turbulence was modeled using k-ϵ turbulence model. The aim of the present study is to investigate the turbulent flow in a duct with varying the arrangements and dimensions of the baffle plates. The obtained computed results show that the boundary layer separation and recirculation regions are significantly affected with the height, thickness and arrangements of the baffle plates. Also the results demonstrate that in the presence of the baffle, the heat is significantly enhanced.

L. C. Demartini investigates the turbulent flow of air inside a channel of rectangular section, containing two rectangular baffle plates. The differential equations that describe the flow were integrated by the Finite Volumes Method, in two dimensions, employing the fluent software with the k-ε model to describe the turbulence. The most important features observed are the high pressure regions formed upstream of both baffle plates, and the extent of the low pressure regions on the downstream regions. The latter are strongly associated with the boundary layer separation on the tip of the baffle plates, which is also influenced by the thickness of the baffle plates

Shivani T. Gajusingh investigates the impact of a rectangular baffle inside a square channel. The measurements were conducted for two Reynolds numbers in the fully turbulent regime. The changes to the flow structure due to the insertion of a baffle were quantified by a direct comparison with the flow structure in the absence of a baffle, under similar conditions. The baffle was attached to the bottom wall of the channel and it obstructed almost 15% of the lower cross-section of the channel, the results show that it influenced the flow structure throughout the channel. Results show that the turbulent velocities are enhanced by a factor of two to three and the rates of energy production and dissipation are enhanced by more than an order of magnitude when a baffle is inserted in the channel. Significant enhancement of turbulence was observed in a region up to two times the baffle height immediately downstream of the baffle. The turbulence in this region was several times higher than that without a baffle.

Nasiuriddin investigates the heat transfer enhancement in a heat exchanger tube by installing a baffle is reported. Three different baffle arrangements were considered. The results show that for the vertical baffle, an increase in the baffle height causes a substantial increase in the Nusselt number but the pressure loss is also very significant. For the inclined baffles, the results show that the Nusselt number enhancement is almost independent of the baffle inclination angle, with the maximum and average Nusselt number 120% and 70% higher than that for the case of no baffle, respectively. For a given baffle geometry, the Nusselt number enhancement is increased by more than a factor of two as the Reynolds number decreased from 20,000 to 5000. Simulations were conducted by introducing another baffle to enhance heat transfer. The results show that the average Nusselt number for the two baffles case is 20% higher than the one baffle case and 82% higher than the no baffle case. The above results suggest that a significant heat transfer enhancement in a heat exchanger tube can be achieved by introducing a baffle inclined towards the downstream side, with the minimum pressure loss.

Kang-Hoon Ko, N.K. Anand[3] have investigated experimentally the average heat transfer coefficient in a rectangular channel which was heated from all the four sides, porous baffles were mounted alternately on the top and bottom walls in staggered manner. Reynolds number was varied between 20,000 and 50,000. The experiment was conducted with three different pore densities (viz.: 10 PPI, 20 PPI, and 40 PPI) and two different thickness (viz.: 1and 0.25in.). Material used for baffle was aluminum foam material. In addition, they conducted an experiment with solid baffle for the sake of comparison. From the study following conclusive statement can be made.

- Heat transfer enhancement was as high as 300% when compared with heat transfer in straight channel with no baffle
- Heat transfer enhancement ratio decreases with increase in Reynolds number and increases with increase in pore density, thickness and height of the baffle.
- The frictional factor decreases with increase in Reynolds number and increase in pore density, thickness and height of the baffle.
- Porous baffle with 40PPI performs the best but also has the highest friction factor.

Athanasia Kalpakli 2012 investigates the turbulent flows in 90 degree curved pipes of circular cross section. The flow cases investigated experimentally are turbulent flow with and without an additional motion, swirling or pulsating, superposed on the primary flow. The aim is to investigate these complex flows in detail both in terms of statistical quantities as well as vortical structures that are apparent when curvature is present. Such a flow field can contain strong secondary flow in a plane normal to the main flow direction as well as reverse flow. The motivation of the study has mainly been the presence of highly pulsating turbulent flow through complex geometries, including sharp bends, in the gas exchange system of Internal Combustion Engines (ICE). On the other hand, the industrial relevance and importance of the other type of flows were not underestimated.

2. Geometric and Mathematical Formulation

The geometry of the problem is presented in Fig. 1-a, b and c. The system consists of air flow moving through a smooth rectangular channel and provided with two and four baffles in three different cases. The rectangular baffle is used. The flow is assumed to be steady and turbulent. In this numerical investigation, the following hypotheses are adopted:

(i) Physical properties of air are constant.
(ii) A profile of velocity is uniform at the inlet.
(iii) The radiation heat transfer is negligible.
(iv) The flow is assumed to be steady.

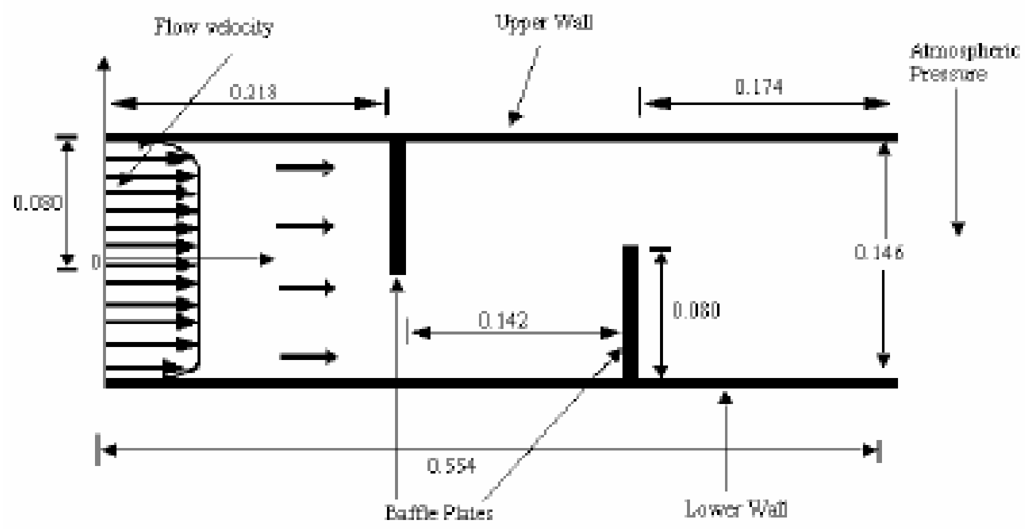


Fig: 1a Detail of the test section with two baffle plates and boundary conditions (dimensions in m).

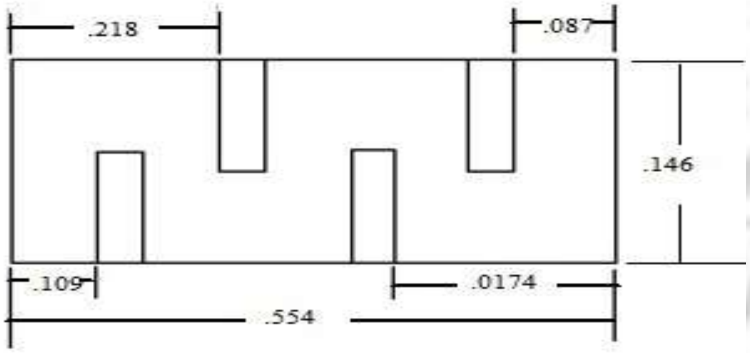
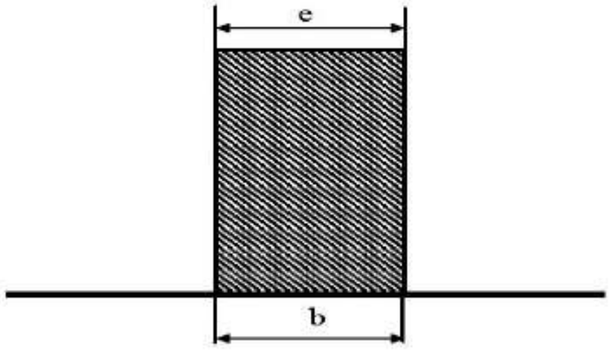


Fig: 1b Detail of the test section with the baffle plates and boundary conditions



(e/b = 1) where e = 0.008

Fig: 1c Specification of baffle plate

2.1 Governing Equations

The governing two-dimensional equations, in a Cartesian coordinate system, for incompressible, steady, with constant fluid properties, are as follows:

I. Continuity equation

$$\frac{\partial U}{\partial X} + \frac{\partial V}{\partial Y} = 0$$

I. Momentum equation

$$\frac{\partial U}{\partial \tau} + \frac{\partial (U^2)}{\partial X} + \frac{\partial (UV)}{\partial Y} = -\frac{\partial P}{\partial X} + \frac{1}{\mathrm{Re}}\left(\frac{\partial^2 U}{\partial X^2} + \frac{\partial^2 U}{\partial Y^2}\right)$$

$$\frac{\partial V}{\partial \tau} + \frac{\partial (UV)}{\partial X} + \frac{\partial (V^2)}{\partial Y} = -\frac{\partial P}{\partial X} + \frac{1}{\mathrm{Re}}\left(\frac{\partial^2 V}{\partial X^2} + \frac{\partial^2 V}{\partial Y^2}\right)$$

II. Energy equation

$$\frac{\partial \theta}{\partial \tau} + \frac{\partial U\theta}{\partial X} + \frac{\partial V\theta}{\partial Y} = \frac{1}{\mathrm{Re}\,\mathrm{Pr}}\left(\frac{\partial^2 \theta}{\partial X^2} + \frac{\partial^2 \theta}{\partial Y^2}\right)$$

The solution domain of the considered two dimensional flows is geometrically simple, which is a rectangle on the x – y plane, enclosed by the inlet, outlet and wall boundaries. The working fluid is air. The inlet temperature of air is considered to be uniform at 300 K. No-slip boundary conditions are taken for the rectangular baffle. Aluminum is selected as the material for baffle. The properties of air taken are standard.

Density(ρ) kg/m^3	Specific heat(c_p) J/kg-k	Thermal conductivity(k) W/m-k	Viscosity(μ) Kg/m-s
1.225	1.0006.43	.0242	1.7894e-05

2.2 Boundary conditions

A fully developed turbulent flow is considered. The quantities U, k , ε are obtained by using numerical calculations based on the k −ε model for low Reynolds Number.
The boundary conditions are listed below:

1- At the inlet of the channel:
$U = U_{in}$, $v = 0$

$K_{in} = 0.005\ U^2in$
$\varepsilon_{in} = 0.1\ k_{in}$

k_{in} stands for the admission condition for turbulent kinetic energy, and ε_{in} is the inlet condition for dissipation.

2- At the walls:
$u = v = 0$
$k = \varepsilon = 0$

3- At the exit: all gradients are null

$$\frac{\partial u}{\partial x} = \frac{\partial v}{\partial x} = \frac{\partial k}{\partial x} = \frac{\partial \varepsilon}{\partial x} = 0$$

$P = P_{atm}$

The Reynolds number based on hydraulic diameter DH is taken according to the experiment of (Endres et al. 2001), it is equal to Re = 8.73 × 104. This a dimensional parameter is defined as follows

$$\mathrm{Re} = \frac{\rho D_h U_O}{\mu}$$

3. Turbulence Model

One of the most widely spread models is the standard k-ϵ model proposed by Launder and Spalding. This model implies two transport equations i.e. turbulent kinetic energy and the dissipation of turbulent kinetic, as follows:

Transport Equation for Turbulent Kinetic Energy k

$$\frac{\partial(\rho k)}{\partial t} + div(\rho k\mathbf{U}) = div\left(-\overline{p'\mathbf{u}'} + 2\mu\overline{\mathbf{u}'e_{ij}'} - \rho\tfrac{1}{2}\overline{u_i'.u_i'u_j'}\right) - 2\mu\overline{e_{ij}'.e_{ij}'} + (-\rho\overline{u_i'u_j'}.E_{ij})$$

Transport Equation for Turbulent Dissipation Rate ε

$$\frac{\partial(\rho\varepsilon)}{\partial t} + div(\rho\varepsilon\mathbf{U}) = div\left[\frac{\mu_t}{\sigma_\varepsilon} grad\ \varepsilon\right] + C_{1\varepsilon}\frac{\varepsilon}{k}2\mu_t E_{ij}.E_{ij} - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}$$

and the eddy viscosity is define as:

$$\mu_t = \rho c_\mu \frac{k^2}{\varepsilon}$$

The model coefficients are (σ_k; σ_ε; $C_{1\epsilon}$; $C_{2\varepsilon}$; $C\mu$) as follows:

C_μ	$C_{1\varepsilon}$	$C_{2\varepsilon}$	σ_k	σ_ε
0.09	1.44	1.92	1.00	1.30

4. Numerical Procedure

The CFD software (Fluent) is used to simulate the fluid flow. The required mesh for computational domain is generated with the help of FLUENT mesh tool. The domain is discretized and equations are formulated using finite volume method. The finite difference governing equations are discretized using the finite volume method. The SIMPLE algorithm is used for the convective terms in the solution equations. The second order up-winding scheme is used to calculate the flow variables. The under relaxation factor is varied between 0.3 and 1.0. The residuals for continuity, momentum and energy equations are all taken as 10-7. The solver iterates the equations till the convergence is obtained for the set residuals.

5. Result and Discussion

The results reported in H. Benzenine et al. work were achieved experimentally. H. Benzenine et al. considered a flat plate baffle. In this work, we investigate the effect of the shape of the baffle on the flow patterns. A waved baffle is considered, and all the modeling conditions are inspired from the experimental study. The numerical model validation is presented in this section.

5.1 Validation of coefficient of friction along the top wall of the channel the rectangular baffles

For the numerical simulations presented in this work, we refer to the numerical work done by (H. Benzenine et al.), who studied the two baffles with a Plane shape and validate with the experimental work of L. C. Demartini.

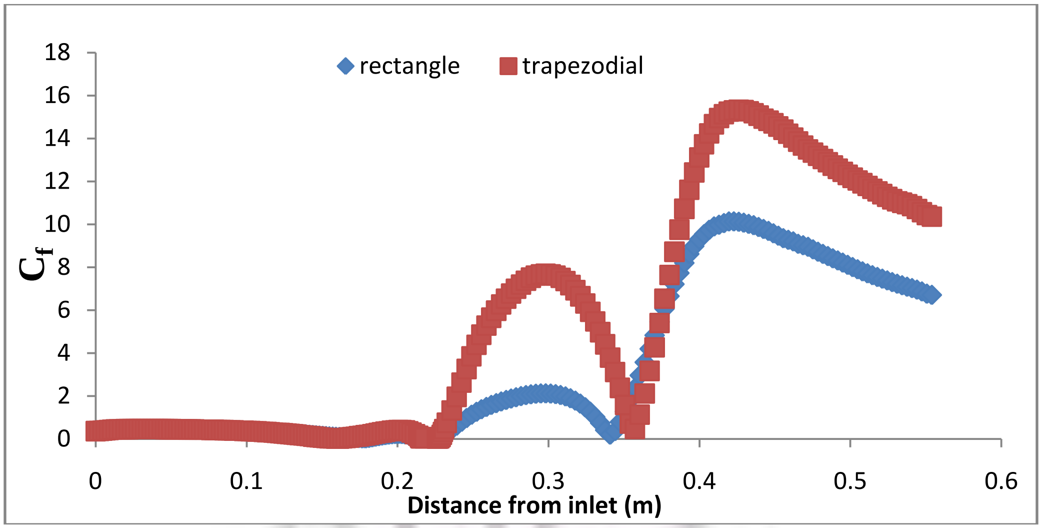


Fig. 2: Variation of coefficient of friction along the higher wall of the channel the two types of baffles

In general, the increase in heat transfer is concerned with the penalty in the terms of coefficient of friction which induces an increase in the pressure drop. The figure 2 shows the variation of the coefficient of friction along the high wall for the two cases. It is noticed that the highest values is in the intermediate zone between the two baffles because the recirculation of the fluid and at the exit.

5.2 Flow characteristics

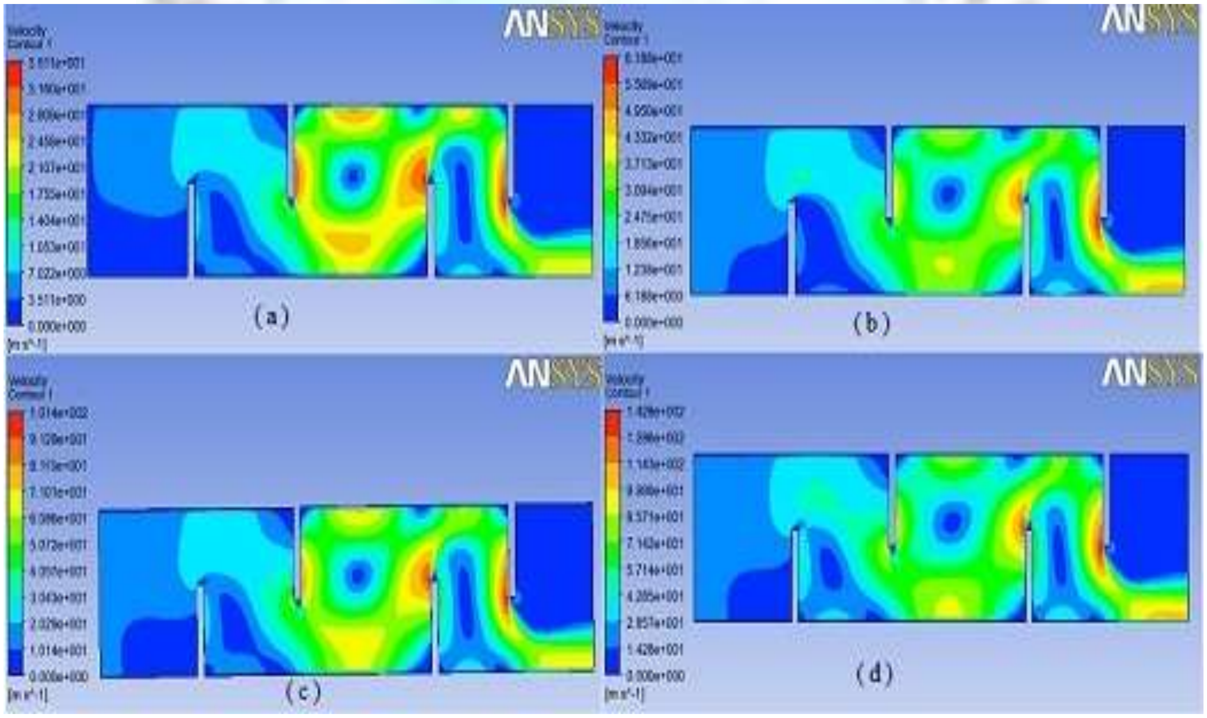


Fig. 3: Axial velocity field in channel for four baffles at Re (a) 4 e4 (b) 8 e4 (c) 12 e4 (d) 18 e4

Fig. 3 shows the axial velocity plots for the rectangular channel having four rectangular baffles at different Reynolds number. It is indicates clearly that the values of velocity are very low in the vicinity of the two baffles especially in the areas located downstream. This is due to the presence of the zones of recirculation. One notices also the increase velocity in space between the top of each baffle and the walls of the channel.

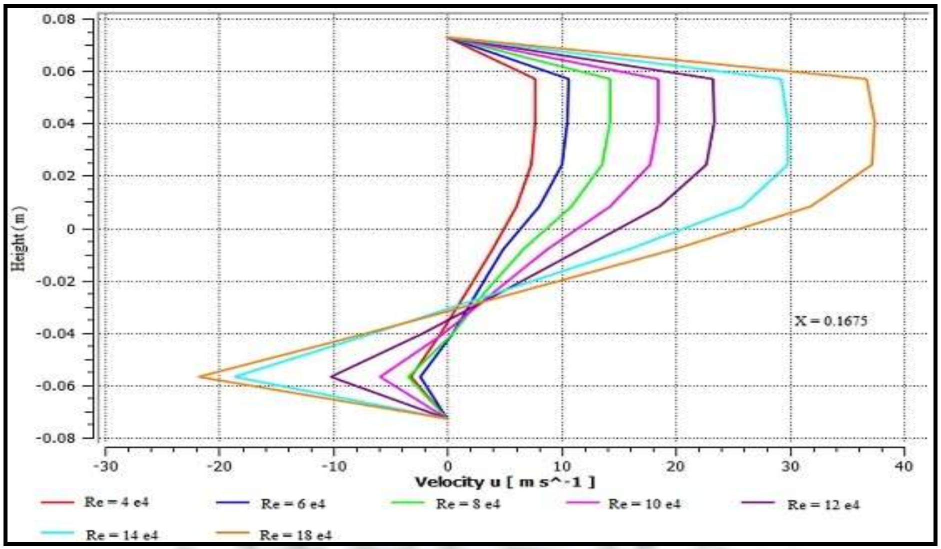


Fig.4: Profiles of axial velocity for x = 0.1675 at different Re

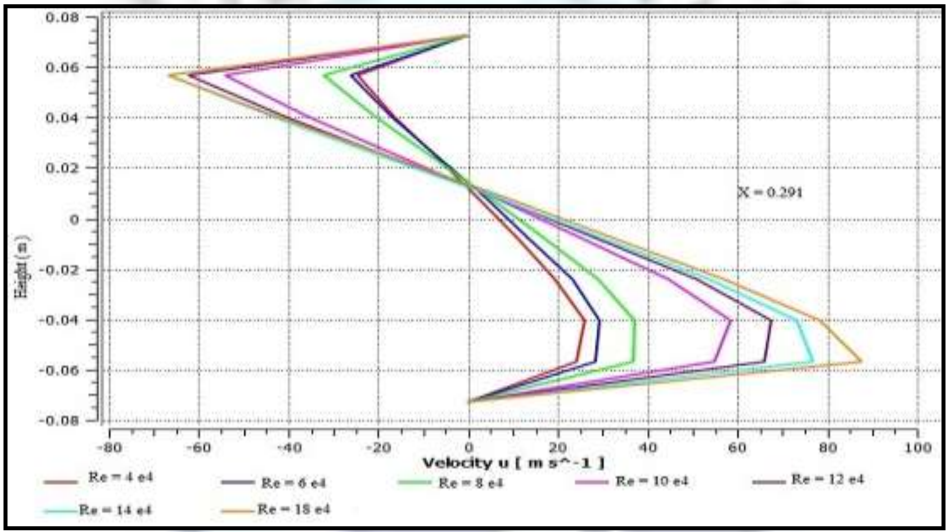


Fig.5: Profiles of axial velocity for x = 0.291 at different Re

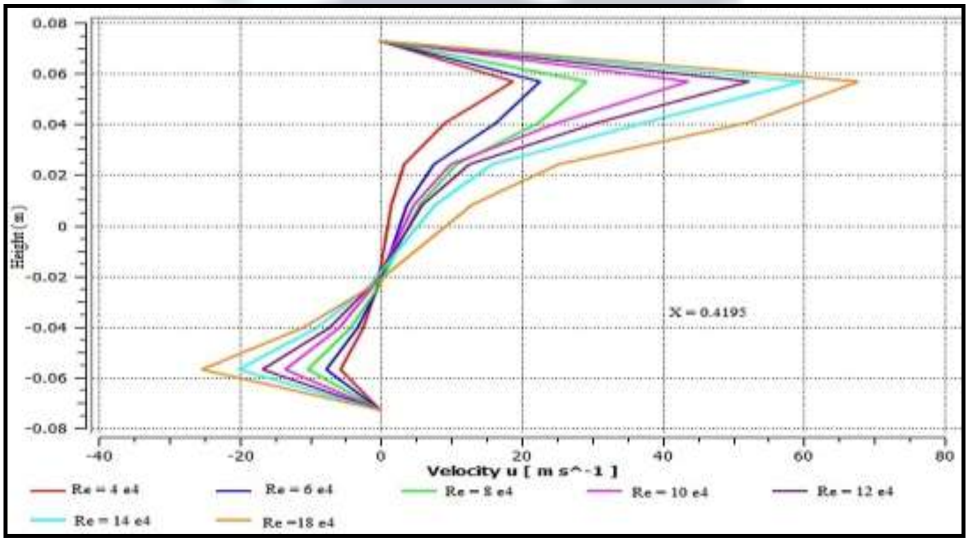


Fig.6: Profiles of axial velocity for x = 0.4195 at different Re

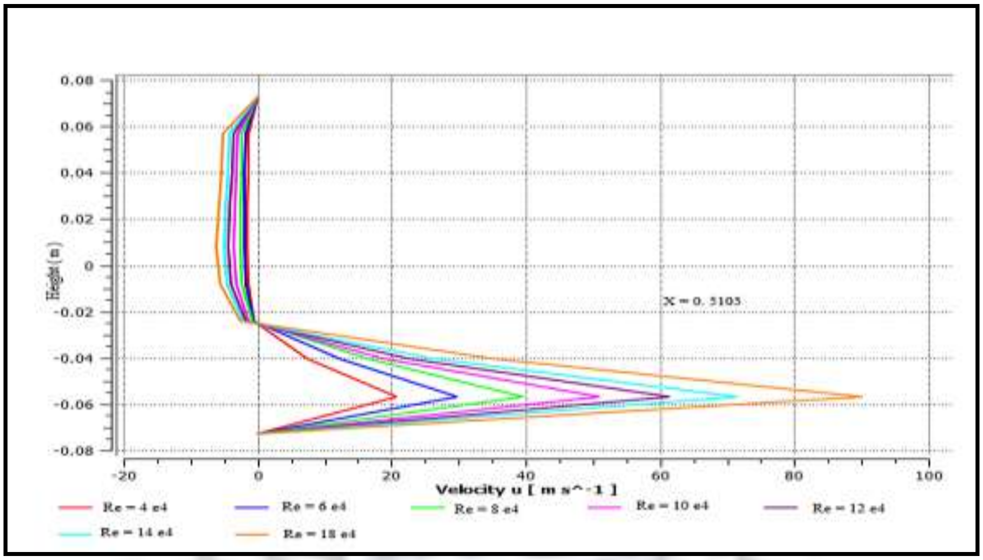


Fig.7: Profiles of axial velocity for x = 0.5105 at different Re

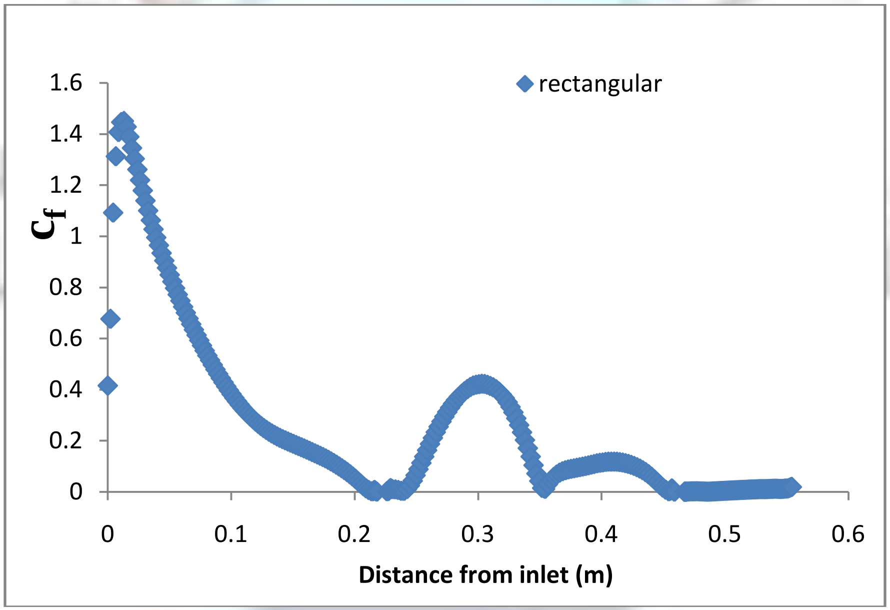


Fig. 8: Variation of coefficient of friction along the higher wall of the channel with four baffles

The figure 8 show that the use of four baffles decrease the coefficient of friction. With the use of four baffles, the coefficient of friction decreases rapidly due to the formation of large vortices near the top of the baffles.

6. Conclusion

The contribution to the study of a turbulent flow in a rectangular channel provided with the four baffles is carried out. The numerical results obtained by the finite volume method, are validated with rectangular chanel having two rectangular baffles and presented to analyze the dynamic behaviour of a turbulent flow using the model k − ε with Reynolds number from 4* 10^4 to 18* 10^4. The evolution axial velocity and the coefficient of friction are treated along the channel and for various Reynolds numbers. The use of the four baffles of rectangular form ensures a considerable decrease in friction coefficient. The velocity increases near the top wall due the acceleration which starts just after the baffle.

7. Nomenclature

C_1 : Constant used in the standard $k-\varepsilon$ model
C_2 : Constant used in the standard $k-\varepsilon$ model
$C\mu$: Constant used in the standard $k-\varepsilon$
e : Width of baffles, m
Uin : Inlet velocity, m/s
Re : Reynolds Number

Greek Symbols

ε :Dissipation rate of turbulence energy, m^2 ρ : Density of the air, kg/m
μe : Effective viscosity, Pa.s ν : Kinematics viscosity, pl

References

[1]. K. Kang-Hoon and N.K. Anand, 'Use of Porous Baffles to Enhance Heat Transfer in a Rectangular Channel', International Journal of Heat and Mass Transfer, Vol. 46, N°22, pp. 4191 - 4199, 2003.
[2]. L.C. Demartini, H.A. Vielmo and S.V. Möller, 'Numeric and Experimental Analysis of the Turbulent Flow through a Channel With Baffle Plates', Journal of the Brazilian Society of Mechanical Sciences and Engineering, Vol. 26, N°2, pp. 153 – 159, 2004.
[3]. Asgari Kaji, Mohammad H., "Turbulent Structure in Open Channel Flow" (2013). Electronic Theses and Dissertations. Paper 4887
[4]. Shivani T. Gajusingh a, Nasiruddin Shaikh a, Kamran Siddiqui a,b,*, Experimental Thermal and Fluid Science 34 (2010) 590–602
[5]. Nasiruddin, M.H. Kamran Siddiqui *, International Journal of Heat and Fluid Flow 28 (2007) 318–328
[6]. H. Benzenine1*, R. Saim2, S. Abboudi3 and O. Imine1, Revue des Energies Renouvelables Vol. 13 N°4 (2010) 639 – 651
[7]. Athanasia Kalpakli 2012, Experimental study of turbulent flows through pipe bends CCGEx & Linn´e Flow Centre, KTH Mechanics, Royal Institute of Technology SE–100 44 Stockholm, Sweden
[8]. Qasim S. Mehdi and Khudheyer S. Mushatet, (2008) Simulation of Turbulent Flow and Heat Transfer Through a Duct with Baffle Plates Journal of Engineering and Development, Vol. 12, No. 3, September (2008)
[9]. Irfan Kurtbas "The effect of different inlet conditions of air in a rectangular channel on convection heat transfer: Turbulence flow", Experimental Thermal and Fluid Science 33 (2008) 140–152