ISSN 0975-3303
Mapana J Sci, **11**, 3(2012), 169-178
https://doi.org/10.12725/mjs.22.11

A Numerical Simulation of Turbulent Flow through a Curved Duct

A K Biswas*, Ashoke K Raman† and A N Mullick‡

Abstract

This paper presents the comparison the results of an experimental work with a numerical work keeping the geometry of the test duct and inlet boundary conditions unaltered. The numerical simulation is validated with the experimental results based on the wall y+ approach for different turbulence models suited for this type of geometry. The experimental work is carried out at mass averaged mean velocity of 40m/s with the measurement of total pressure by a pre-calibrated multi-hole pressure probe and the results presented in the form of a pressure contours in 2-D. For validation of the numerical results Standard k-ε, k-ω and Reynolds Stress Model (RSM) are used to solve the closure problem. The turbulence models are investigated in the commercial CFD code of Fluent using y+ value as guidance in selecting the appropriate grid configuration and turbulence model. Based on the wall y+ values for different turbulence models, it is concluded in the present study that the mesh resolving the fully turbulent region is sufficiently accurate in terms of qualitative features and RSM turbulence model

* Department of Mechanical Engineering, National Institute of Technology, Durgapur 713209, WB, India

† Department of Mechanical Engineering, National Institute of Technology, Durgapur 713209, WB, India

‡ Department of Mechanical Engineering, National Institute of Technology, Durgapur 713209, West Bengal, India. amaranth.mullick@me.nitdgp.ac.in

Received: July 2012, Reviewed: Aug. 2012

predicts the best results while comparing with the experimental results.

1. Introduction

Curved ducts are used in aircraft intakes, combustors, internal cooling system of gas turbines, ventilation ducts, wind tunnels etc. Heat exchangers in the form of curved ducts are used widely in food processing, refrigeration and hydrocarbon industries. Gas turbine engine components such as turbine compressors, nozzle etc. utilise several complex duct configuration. Performance of duct flow depends upon the geometrical and dynamical parameters of the duct. So it is very much essential to design the duct with proper geometry to improve the performance. Study of flow characteristics through constant area ducts is a fundamental research area of basic fluid mechanics since the concepts of potential flow and frictional losses in conduit flow were established. Duct is a part and parcel of any fluid-mechanical system. It is a passageway made of sheet metal or other suitable material used for conveying air or other gases or liquids at different pressures. Depending on its application the shape the duct may be either of straight, curved, annular, polar, sector, trapezoidal, rhombic etc. Flow through curved ducts has practical importance in chemical and mechanical industries in particular. Obviously, compared to a straight duct, flow in a curved duct is more complex due to curvature of the duct axis. It induces centrifugal forces on the flowing fluid resulting in the development of a secondary motion (normal to primary direction of flow) which is manifested in the form of a pair of counter-rotating vortices. Depending on the objective, fluid mechanical systems often demands for the design of ducts with complex geometry (like inlets, nozzles, diffusers, contractions, elbows etc) albeit with high efficiency. In these applications, design of the ducts is based on the mathematical formulation of the flow field for the prescribed condition. This paper presents a part of the work which involves the investigation of the flow parameters like mean velocity, static pressure, total pressure, coefficients of pressure recovery and pressure loss at different Reynolds numbers and comparing the results with similar intake ducts of constantly increased area.

However, due to the space constraint only the wall pressure distribution is presented here.

Rowe [1], carried out experiments on circular 90° and 180° turn curved ducts with Re=0.4x10E+5 and reported the generation of contra rotating vortices within the bends. Bansod & Bradshaw [2], studied the flow characteristics within the 22.5°/22.5° S-shaped constant area ducts of different lengths and radii of curvature. They reported the development of a pair of contra-rotating vortices in the low pressure zone at the exit of the duct which was the consequence of the effect of stream wise vortices developed in the first half of the duct. Enayet et al. [3], investigated the turbulent flow characteristics through 90° circular curved duct of curvature ratio 2.8. It was observed that the thickness of the inlet boundary layer has a significant role on generation of secondary motion within the duct. Azzola et al.[4], have studied the turbulent flow characteristics through 180° circular bend with curvature ratio of 3.375 through experiments as well as computational methods. They observed a pair of contra-rotating vortices arising out of secondary motion in both experimental and numerical studies. Lacovides et al., [5], reported the flow prediction within 90° curved duct using numerical simulations based on the experimental investigation by Taylor et al.[6]. They adopted finite volume approach to solve the semi-elliptical form of equation for 3-D flow analysis considering the wall function in the region close to the wall. The result shows a good agreement between the experimental and numerical analysis. Thangam and Hur [7], studied the secondary flow of an incompressible viscous fluid in a curved rectangular duct by using a finite volume method. They reported that with the increase of Dean Number the secondary flow structure evolves into a double vortex pair for low aspect ratio ducts. They correlated friction factor as a function of the Dean Number and aspect ratio. Kim and Patel [8], have investigated on a 90° curved duct of rectangular cross-section with aspect ratio 6 using five-hole probe and cross-wire hot wire anemometer. They reported the formation of vortices on inner wall due to the pressure driven secondary motion originated in the corner region of curved duct. Investigation on the turbulent boundary layer on the wall of an S-shaped wind tunnel for various Reynolds numbers ranging from $3.0x10^3$ to $11.0x10^3$ was carried out by Burns et al. [9]. They used hot wire probe to measure mean velocity and Reynolds stresses. They interpreted their results for turbulence response and evaluated Reynolds Stress Transport Equations. Singh et al. [10], experimentally studied the flow and performance characteristics of a Y-shaped duct having an aspect ratio 1 and 1.66 for two inlet limbs with angle of turn 90°/90°. The average inlet velocities in the two limbs were 29m/s and 24m/s respectively. The longitudinal velocity and static and total pressure were measured by using a 3-hole pressure probe. They observed that the pressure recovery coefficient and loss coefficient increased continuously from inlet to the exit of the diffusing duct and are nearly same. From the available literature on curved ducts it is apparent that the studies are generally related to straight or curved ducts with circular cross-section. The present experimental investigation aims for a systematic study on the flow pattern of a curved C shaped duct through the measurements of wall pressure distribution, and selecting the best turbulence model for validation of the experimental results.

2. Experimental Set up

Experiment is carried out using the facility of wind tunnel at the Aerodynamics Laboratory of National Institute of Technology, Durgapur. Schematic layout of the experimental set-up is shown in Fig.1. A centrifugal air blower is directly coupled with a three phase, 5.5 kW electric motor of speed 2870 rpm. In order to minimize the vibration transmitted by the air supply unit to the test-piece, a flexible extension made of canvas of dimension 0.155mx0.310m is fitted at the blower outlet. The blower is followed by conical diffuser made of G.I. sheet of length 1.38m having inlet and outlet diameters of 0.310m and0.600m respectively with a diverging angle of 6.66°. The settling chamber is a straight cylindrical duct through which the discharge from the conical diffuser flows. It is of uniform diameter of 0.600m and length 2.88m. Nylon screens are provided at three locations in the settling chamber in the transverse direction of flow for straightening as well as reducing the turbulence level of the flow. The contraction piece is installed between the settling chamber and the inlet piece of the curved duct. The contraction piece is of hollow truncated pyramid shape made of ply wood symmetric about the centre line with rectangular sections at both of its inlet and outlet. The inlet dimension of the contraction piece is 0.30mx0.30m and outlet dimension is 0.05mx0.10m. This piece when connected to the duct with a straight extension piece ensures uniform velocity profile at the inlet of the duct as well as reduces the resultant turbulence level at its own exit section. The complete geometry of curved duct under test is shown in fig.2. It is a rectangular 90°curved duct of width 50mm (W) and height 100mm (b) with a centre line length of 600mm (L). It is constituted of four equal segments each subtending at an angle of 22.5°. The parallel horizontal walls (top and bottom) of the duct are made from 12mm thick transparent Perspex sheet whereas the curved vertical walls (convex and concave) are fabricated with 3mm thick Perspex sheet. These side walls of the duct are made by bending the sheet and fastened by screws with the top and bottom parallel walls. The radii of curvature of the outer and inner curved walls of the duct are 407mm 357mm respectively. The mean radius of curvature of the duct is 382mm (Rc). Two straight constant area ducts of cross-sectional area 50mmx100mm were attached as extension pieces at the inlet and exit respectively. Length of these two extension pieces are 100mm. They help fixing the inlet and outlet conditions of the flow. There are six sections considered at the middle point of all these six pieces of rectangular curved duct. These sections are inlet-section, section-A, section-B, section-C, section-D and outlet-section as shown in Fig.2. These six rectangular sections are separately shown in Fig.3. Wall static pressure on different walls of the duct was measured by using the wall static pressure tapping. There are five holes at a distance 5mm, 15mm, 25mm, 35mm and 45mm from the edge on the top and bottom faces. Similarly on the inside and outside curved faces there are nine holes at a distance 5mm, 15mm, 30mm, 40mm, 50mm, 60mm, 70mm, 85mm and 95mm from the edge. All the holes are drilled with a diameter of 2mm. Hollow stainless steel tubes of length 20mm to 25mm are inserted into the holes such that tubes ends are not projected beyond the inside surface of the walls of the duct. The tubes are fitted into the hole by using adhesive available in the market. For recording any pressure measurement, a particular tap is connected to the inclined tube manometer through flexible tube while all other tapping are plugged by caps, the other limb of differential manometer are kept open to atmosphere. A multi-tube inclined (35° with vertical) manometer has been used to measure the pressure head at different points simultaneously. Kerosene oil is used as manometric liquid. The velocity of air was measured by inserting a Pitot tube at the midpoint of the exit section of the duct. Experiment is carried out for three different air velocities of rather low magnitudes of 20m/s, 40m/s and 60m/s to ensure incompressible flow condition at low Mach number.

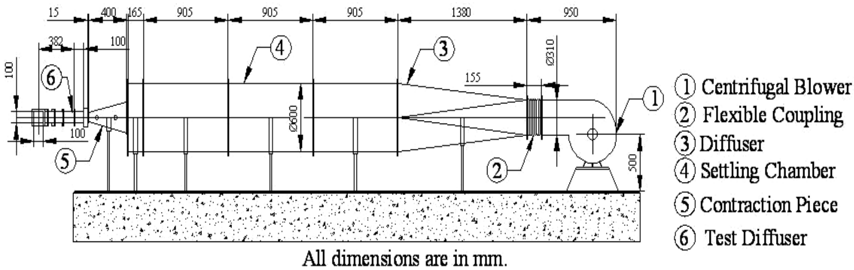


Fig. 1: Schematic Diagram of Wind Tunnel with Test Diffuser

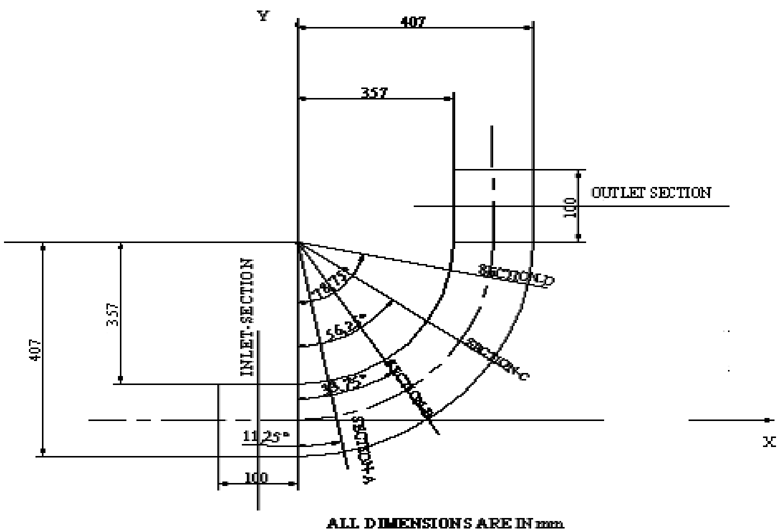


Fig. 2: Geometry of 90° Curved Duct and measuring locations

3. Computational Methodology and Boundary Conditions for C Duct

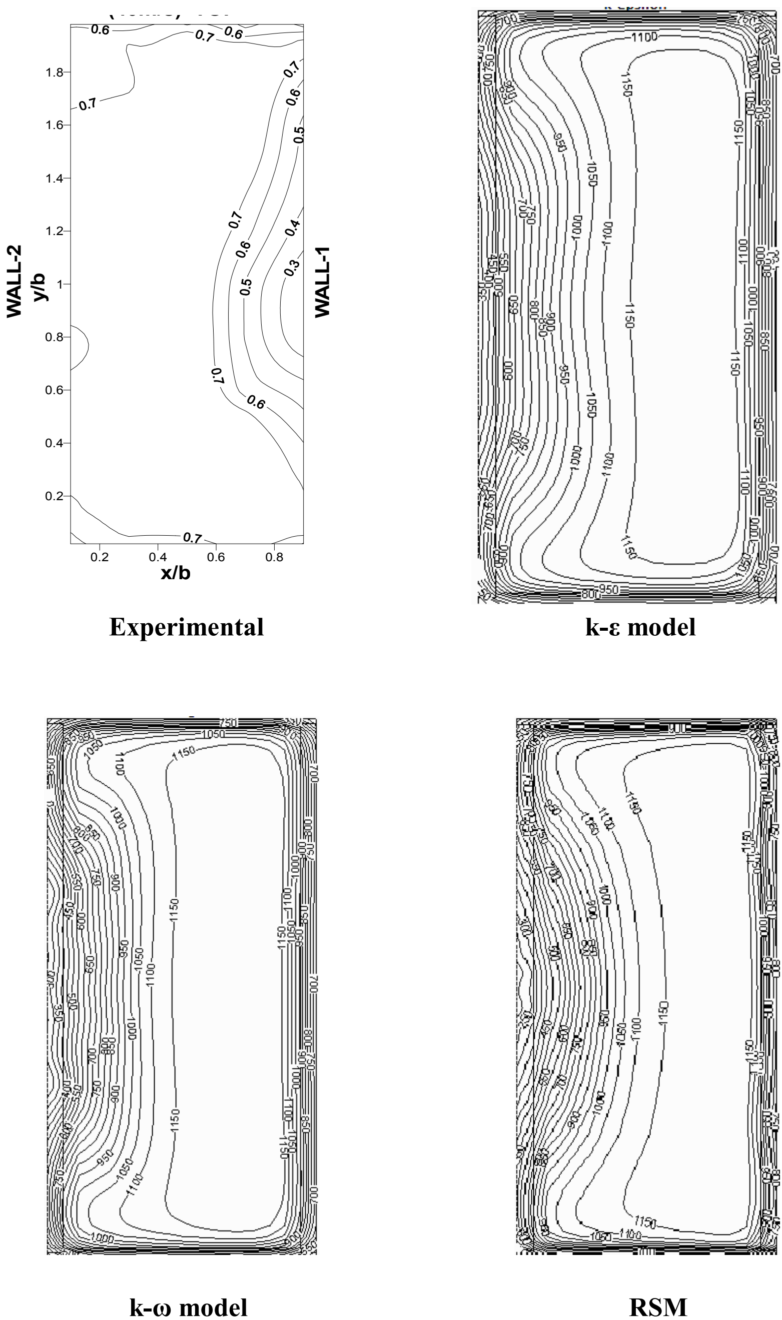
The pre-processor GAMBIT is used to create the geometry defining the problem and discretize the domain while FLUENT 6.9 is employed to discretize and solve the governing equations. The generated mesh size was of 10^5 cells. Three boundary conditions where specified, i.e. velocity inlet, pressure outlet and wall. No slip boundary conditions were applied to the walls of the duct. For velocity inlet boundary condition the turbulence intensity was taken as 10% and the hydraulic diameter was taken as 66.67mm where as for pressure outlet boundary condition the backflow modified turbulent viscosity was taken as 0.001 m^2/s.

Table 1

Turbulence Model	Y+ (minimum)	Y+(maximum)
Standard k-ε	24.41	160.29
k-ω	24.75	154.80
RSM	20.61	162.16

4. Results and Discussions

In the present work the numerical work is compared with experimental work under various turbulence models. Here, Standard k-ε, k-ω and RSM models are attempted. Based on the wall y+ values, it has been observed that the RSM model provides the best result both at the viscous blending region and fully turbulent region. Moreover the RSM model shows a good qualitative matching with the experimental work. It is also evident from both experimental and numerical work that the bulk fluid shifts towards the outer side of the curved duct.



Experimental

k-ε model

k-ω model

RSM

Fig. 3: Total Pressure contours at the outlet section of the C duct

5. Conclusions

i. From the experimental work it is observed that the low momentum fluid shifted towards the inner wall of the curved duct.

ii. The RSM turbulence model predicts the results best comparing the other models.

References

[1] M Rowe, Measurements and computations flow in pipe bends, *J. Fluid Mechanics*, vol. 43, pp. 771-783, 1970.

[2] P Bansod and P Bradshaw, The flow in S-shaped ducts, *Aeronautical Quarterly*, vol. 23, pp. 131-140, 1972.

[3] M M Enayet, M M Gibson, A M K P Taylor and M Yianneskis, Laser doppler measurements of laminar and turbulent flow in a bend, *Int. J. Heat and Fluid Flow*, vol. 3, pp. 211-217, 1982.

[4] J Azzola, A C Humphrey, H Iacovides and B E Launder, Developing turbulent flow in a U-bend circular cross-section: Measurement and computations, *Trans. ASME: J. Fluid Engineering*, vol. 108, pp. 214-221, 1986.

[5] H Lacovides, B E Launder and P A Loizou, Numerical computation of turbulent flow through a squared sectioned 90° bend, *Int. Journal of Heat and Fluid Flow*, vol. 8, pp. 320-325, 1987.

[6] A M K P Taylor, J H Whitelaw and M Yianneskis, Curved ducts with strong secondary motion: Velocity measurements of developing laminar and turbulent flow, *Trans. ASME: J Fluid Engineering*, vol. 104, pp. 350-358, 1982.

[7] S Thangam and N Hur, Laminar secondary flows in curved rectangular ducts, *J Fluid Mechanics*, vol. 217, pp. 421-440, 1990.

[8] W J Kim and V C Patel, Origin and decay of longitudinal vortices in the development of flow in a curved rectangular duct, *Trans. ASME, J. Fluid Engineering*, vol. 116, pp. 45-52, 1994.

[9] J M Burns, H H Fernholz and P A Mankewitz, An experimental investigation of a three dimensional turbulent boundary layer in an S-shaped duct, *J. Fluid Mechanics*, vol. 393 , pp. 175-213, 1999.

[10] N Singh, S N Singh and V Seshadri, Flow characteristics of an asymmetric Y-shaped duct, *Proceedings of 31st National Conference on Fluid Mechanics and Fluid Power*, Jadavpur University, pp. 749-757, 2004.