

A Computational Analysis of Flow Structure Through Constant Area S-Duct

¹A. K. Biswas, ²Prasanta K Sinha, ¹A. N. Mullick and ³B. Majumdar

¹National Institute of Technology, Durgapur 713 209, West Bengal, India

²Durgapur Institute of Advanced Technology & Management, West Bengal, India

³Jadavpur University, Kolkata, West Bengal, India

Abstract.

This paper presents the results of an experimental work with measurement of mean velocity contours in 2-D form and validation of the same with numerical results based on the y^+ approach at fully developed flow for various turbulent models like, $k-\epsilon$ model, $k-\omega$ model, RNG $k-\epsilon$ model and Reynolds Stress Model (RSM), are used to solve the problem. All the turbulence models are studied in the commercial CFD code of Fluent. The experiment is carried out at mass averaged mean velocity of 40m/s and the geometry of the duct is chosen as rectangular cross-section of $45^\circ/45^\circ$ curved constant area S-duct. In the present paper the computational results obtained from the different turbulence models are compared with the experimental results. In addition to this for validation of the numerical simulation near wall treatments for fully developed flow or log-law region are also investigated for wall $30 < y^+ < 300$ in the region where turbulent shear dominates. It is concluded from the present study that the mesh resolving the fully turbulent region is sufficiently accurate in terms of qualitative features. Here RSM turbulence model predicts the best results while comparing with the experimental results. RSM model also predicts the flow properties more consistently because it accounts for grid independence test.

Keywords: Constant area S-duct, Turbulence Models, Wall function, Fluent Solver

I. Introduction

Constant area curved ducts are used for many engineering applications like small aircraft intakes, combustors, internal cooling system of gas turbines, HVAC ducting system, wind tunnels, heat exchangers in food processing refrigeration and hydrocarbon industries etc. In order to improve the performance of a duct it is absolutely necessary to design the duct with proper geometry so that the losses due to friction and eddies are minimized. Depending upon its application, the shape of the duct is chosen either straight or curved or annular or polar or sector. As a matter of fact the flow through a curved duct is more complex compared to straight duct due to the curvature of the centerline. It induces centrifugal forces on the flowing fluid resulting in the development of a secondary motion, which is manifested in the form of a pair of contra-rotating vortices. Depending on the objective, hydro-mechanical systems often demand for the design of ducts with complex geometry 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. Rowe [1] carried out experiments on circular 90° and 180° turn curved ducts with $Re=0.4 \times 10^5$ and reported the generation of contra-rotating vortices within the bends. Bansod & Bradshaw [2] studied the flow characteristics

within the $22.5^\circ/22.5^\circ$ 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 that was the consequence of the effect of stream wise vortices developed in the first half of the duct i.e. before the inflexion. 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. Lacovides et al. [4] reported the flow prediction within 90° curved duct using numerical simulations based on the experimental investigation by Taylor et al. [5]. 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 [6] 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 [7] 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. Turbulent flows are highly affected by the presence of solid boundary due to the no-slip condition. Gerasimov [8] emphasized that accurate presentation of the near wall region is paramount to successful simulation of wall-bounded turbulent flow. Salim and Cheah [9] studied the wall y^+ strategy and concluded that wall y^+ is a suitable selection criterion for determining the appropriate mesh configuration and turbulence model, coupled with near wall treatment which leads to the accurate computational prediction in Fluent. Ariff et. al. [10] also studied the wall y^+

strategy and concluded that for fluid problems with complex turbulent flow structures different models for their flow prediction show their best result at different flow region. They also concluded that the near wall regions have larger gradients in the solution variables, and momentum and other scalar transports occur most vigorously. In the investigation of the behavior of turbulence models and near wall-treatment they followed the maximum and minimum values.

(a) $5 < y^+ < 30$ for buffer layer where both viscous and turbulent shear dominates

(b) $30 < y^+ < 300$ for fully turbulent region where turbulent shear predominates

II. Experimental set up

Experiment is carried out at the Aerodynamics Laboratory of National Institute of Technology Durgapur. The schematic diagram of the experimental set up is shown in Fig. 1.

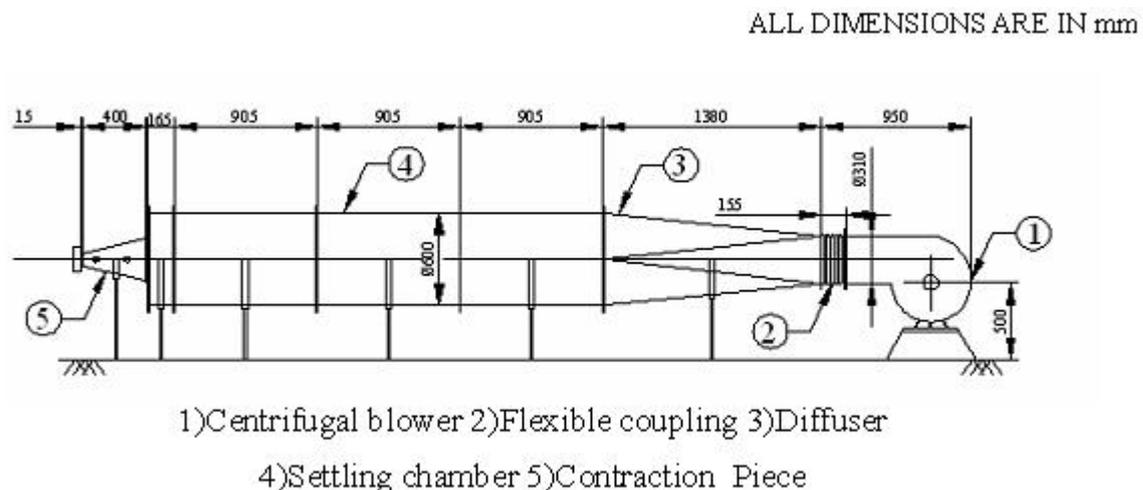


FIGURE 1. Schematic layout of the experimental set up

The experimental set up consists of a wind tunnel, which is driven by an electric motor of 5.5 kW power. The test piece is connected with the settling chamber via a constant area straight duct to ensure uniform velocity profile at the inlet section of the test piece. The geometry of the curved duct under test is shown in Fig. 2. It is a rectangular 90° curved duct of width 50mm wide and 100mm height with a centerline length of 600mm. It is constituted of four equal segments of 22.5° each. The entire test piece is made of Perspex sheet. Two straight constant area ducts of cross sectional area 50mm x 100mm and 100mm long are connected as extension pieces at the inlet and exit of the test piece. Middle points of all the six segments are considered as six sections and they are assigned as Inlet Section, Section - A, Section- B, Section - C, Section- D and Outlet Section.

The mean velocity, static pressure and total pressure are measured with the help of a multi-hole pressure probe. At each section 125 locations are chosen to measure the flow parameters.

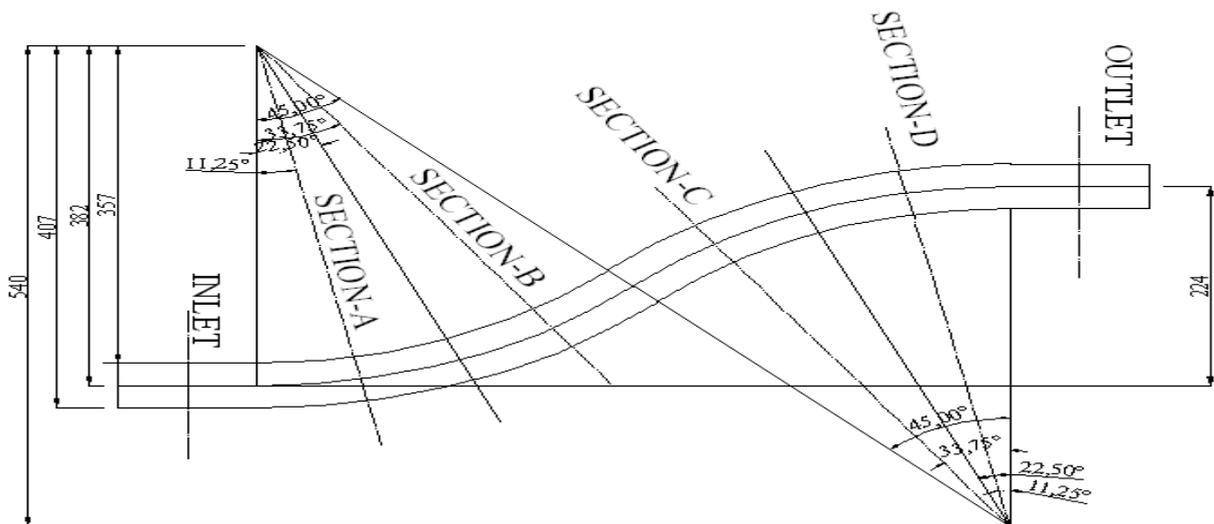


FIGURE 2. Schematic Diagram of the 45°/45° Curved Duct

III. Computational methodology

The wall y^+ is a non-dimensional number similar to local Reynolds number, determining whether the influences in wall adjacent cells are laminar or turbulent, hence indicating the part of the turbulent boundary layer and it is given as

$$y^+ = \frac{u^* y_p}{\nu}$$

Where, $u^* = \sqrt{\frac{\tau_w}{\rho}}$

y^+ on the distance of the centroid of the nearest cell from the wall (y_p) as well as on the wall shear stress (τ_w) for a specification problem. Thus the grid refinement process near the wall is generally done by a trial and error basis.

In the present study 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. Initially the mesh size of 1216000 cells was created. Three boundary conditions were specified, i.e. velocity inlet, pressure outlet and wall. No slip boundary conditions were applied to the walls of the duct. During the creating of the mesh no boundary layer was taken, and entire turbulent zone was taken. A general method for determining the most appropriate mesh configuration is grid independence test, where different meshes are tested until the solution is independent of further refinement, by matching the results to experimental results. The maximum and minimum y^+ values corresponding to various turbulent models are shown in Table 1.

Table 1. Ranges of y^+ values for different turbulent models

| Turbulence Model | Y+(min) | Y+(max) |
|------------------|---------|---------|
| K-epsilon | 45.56 | 97.00 |
| RSM | 45.50 | 99.16 |
| k-omega | 39.68 | 99.78 |
| RNG k-epsilon | 48.65 | 96.68 |

IV. Results and discussion

In the present work an attempt is made to compare the experimental work with the numerical simulation and for this purpose four different turbulent models namely, k-ε model, k-ω model, RNG k-ε model and RSM model. It has been observed that with wall y^+ approach for grid refinement process RSM

model provides the best result among all the four turbulence models. Wall y^+ value at the viscous blending region is lowest and the same at fully turbulent region is highest for the RSM model. Moreover the mean velocity contours at the outlet section drawn with RSM model are in good agreement with the experimental result qualitatively. The comparison of the mean velocity contours in experimental work and numerical work is shown in Fig. 4. Though all the turbulence models are tried out for the validation purpose but due to the space constraint only the RSM model is presented here. From the diagram it is evident that due to formation of secondary motion the bulk velocity shifted towards the outer wall which is an efficacy of the curved ducts.

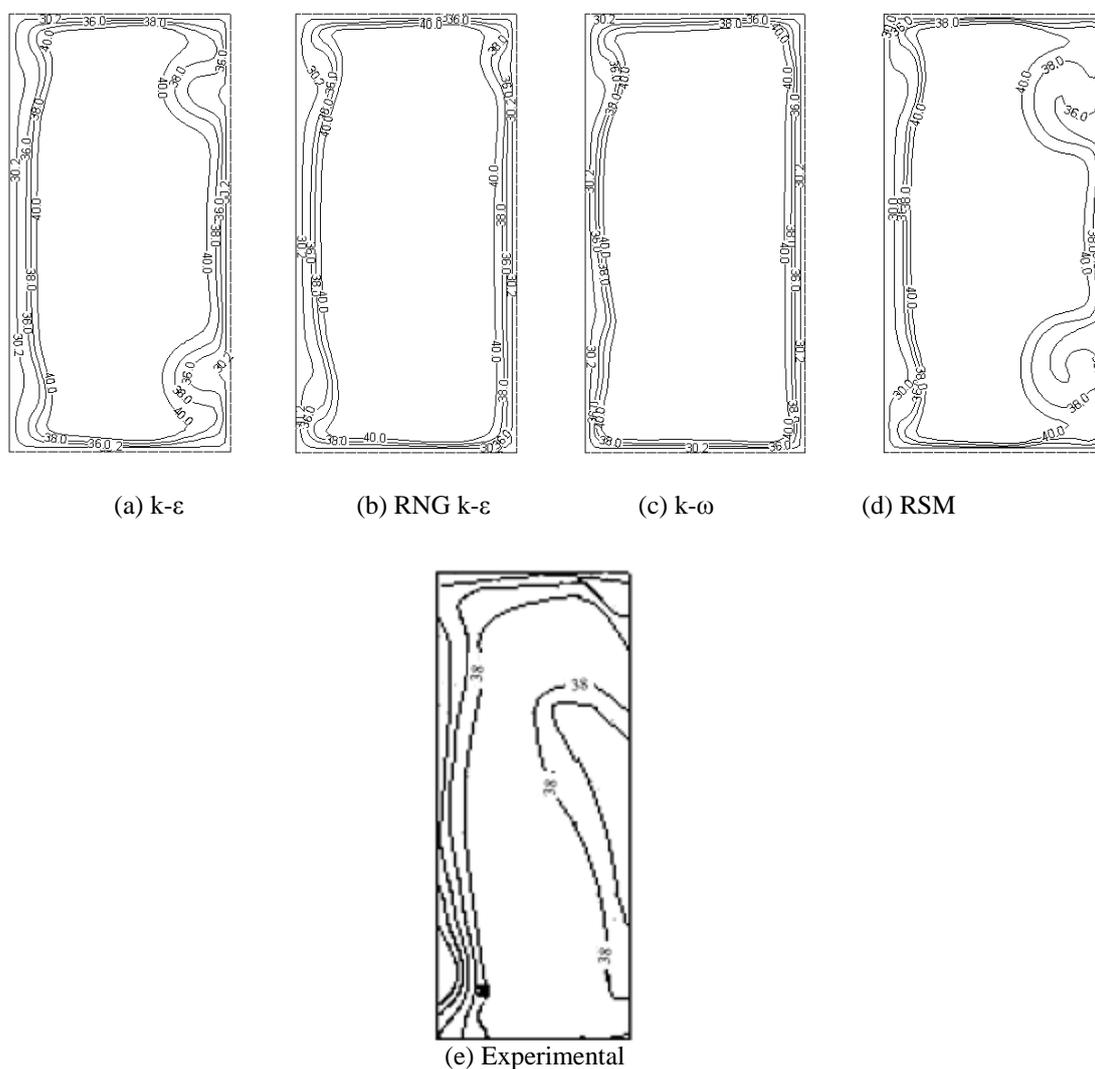


FIGURE 3. Comparison of experimental and numerical results

V. Conclusions

The present study shows that the non-dimensional wall y^+ is a suitable selection criterion for determining the appropriate mesh configuration and turbulence model, coupled with near wall treatment, that lead to accurate computational prediction in Fluent. Moreover both experimental and numerical shows that the high momentum fluids shifted towards the outer wall due to the generation of secondary motion and $k-\epsilon$ turbulent model predicts the same result with a very good agreement with the experimental result.

References

- [1] M. Rowe., "Measurements and computations of flow in pipe bends", *Journal of Fluid Mechanics*, 1970, 43(4), pp.771-783.
- [2] P. Bansod and P. Bradshaw., "The Flow in S shaped Ducts", *Aeronautical Quarterly*, 1972, pp. 23 131-140.
- [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. Journal of Heat and Fluid Flow*, 1982, 3, pp. 211-217.
- [4] 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*. 1987, 8(4), pp. 320-325.
- [5] 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 Journal of Fluid Engineering*. 1982, 104, pp. 350-358.
- [6] S. Thangam and N. Hur. “Laminar secondary flows in curved rectangular ducts”, *Journal of Fluid Mechanics*. 1990, 217, pp. 421-440.
- [7] 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 Journal of Fluid Engineering*. 1994, 116(3), pp. 45-52.
- [8] A. Gerasimov., “Modelling Turbulent flow with Fluent”, Ansys Inc, Europe, 2006.
- [9] S. M. Salim and S. C. Cheah., “Wall Y+ Strategy for dealing with wall-bounded Turbulent Flows”, *Proc. Int. Multiconference of Engg & Comp, Scientists*, 2009, Vol. II.
- [10] M. Ariff, S. M. Salim and S. C. Cheah., “Wall Y⁺ approach for dealing with turbulent flow over a surface mounted cube: Part II – High Reynolds number”, *7th Int. Conf. on CFD in the mineral and process Industries*, Australia, 2009.