RESEARCH ARTICLE OPEN ACCESS

A Computational Analysis of Flow Development through a Constant Area C- Duct

A. K. Biswas[1], Prasanta K Sinha[2*], A. N. Mullick[1] & B Majumdar[3]
[1]*National Institute of Technology, Durgapur, West Bengal, India*
[2]*Durgapur Institute of Advanced Technology& Mangt, Durgapur, West Bengal, India*
[3]*Jadavpur University, Kolkata, West Bengal, India*

Abstract.
This paper represents the results of an experimental work with measurement of mean velocity along with total pressure contours in 2-D form and validation of the same with numerical results based on the wall y^+ approach for various turbulent models like, Spalart Alamras, k-ε model, k-ω model and RSM models are used to solve the closure problem. The turbulence models are investigated in the commercial CFD code of Fluent using y^+ as guidance in selecting the appropriate grid configuration and turbulence model. 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 90 curved constant area duct .In the present paper the computational results obtained from different turbulence models are compared with the experimental result along with the near-wall treatments are investigated for wall $y^+<30$ in the region where both viscous and turbulent shear dominates and $y^+>30$ in the fully turbulent region. It is concluded in 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.
Keywords: Constant area C-duct, Turbulence Models, Wall function, Fluent Solver

I. Introduction

The uses of constant area curved ducts are many in the engineering applications. These are used in small aircraft intakes, combustors, internal cooling system of gas turbines, HAVC ducting system, wind tunnels, heat exchangers in food processing refrigeration and hydrocarbon industries etc. Performance of a curved constant area duct depends on the geometrical and dynamical parameters, so it is absolutely necessary to design the duct with proper geometry in order to minimize the losses due to friction and eddies, which in turn improve the performance of the duct.

Since the concepts of potential flow and frictional losses in internal flow were established, the study of flow characteristics through constant area ducts has earmarked as a fundamental research area of basic mechanics. 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 $R_e=0.4x10^5$ 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 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

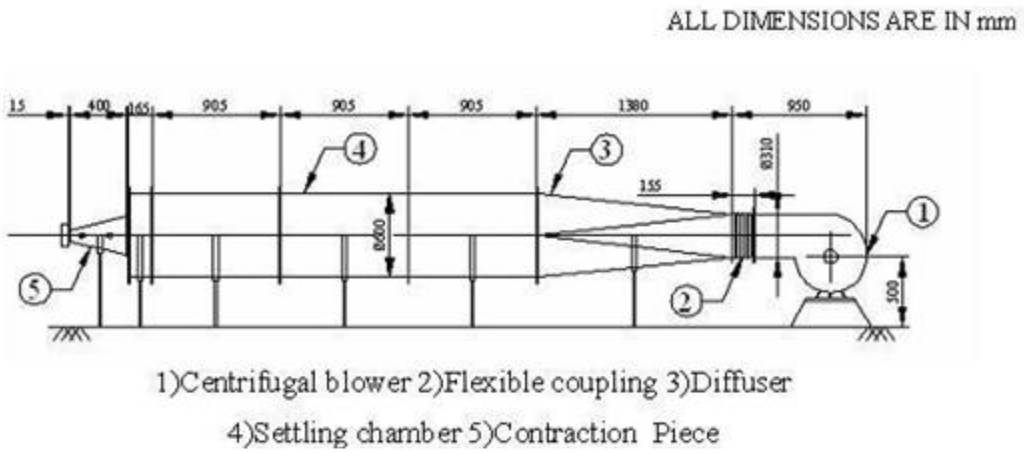


Fig. 1 Schematic layout of the experimental set up

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 prove. At each section 125 locations are chosen to measure the flow parameters.

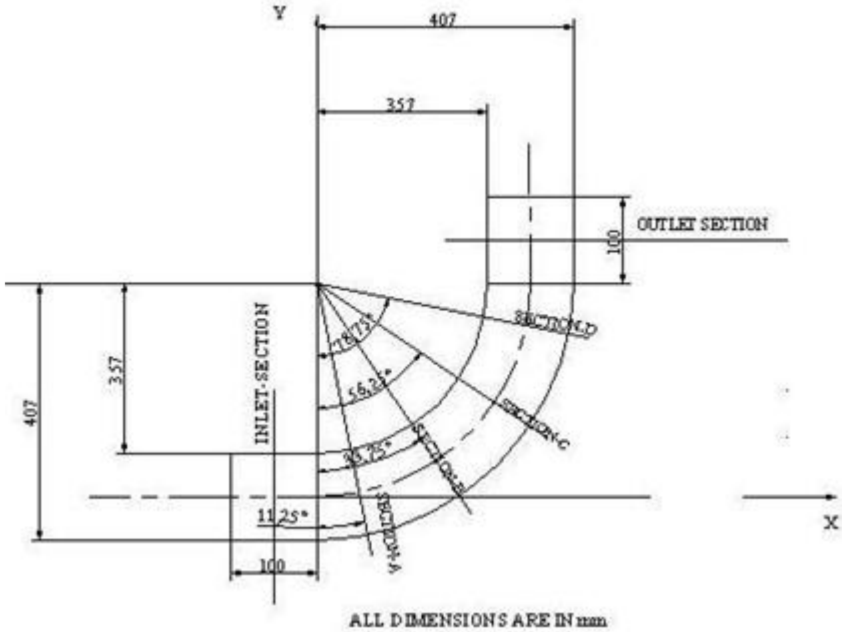


Fig.2. Schematic Diagram of the 90° Curved Duct showing its different plane

III. Computational methology

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

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

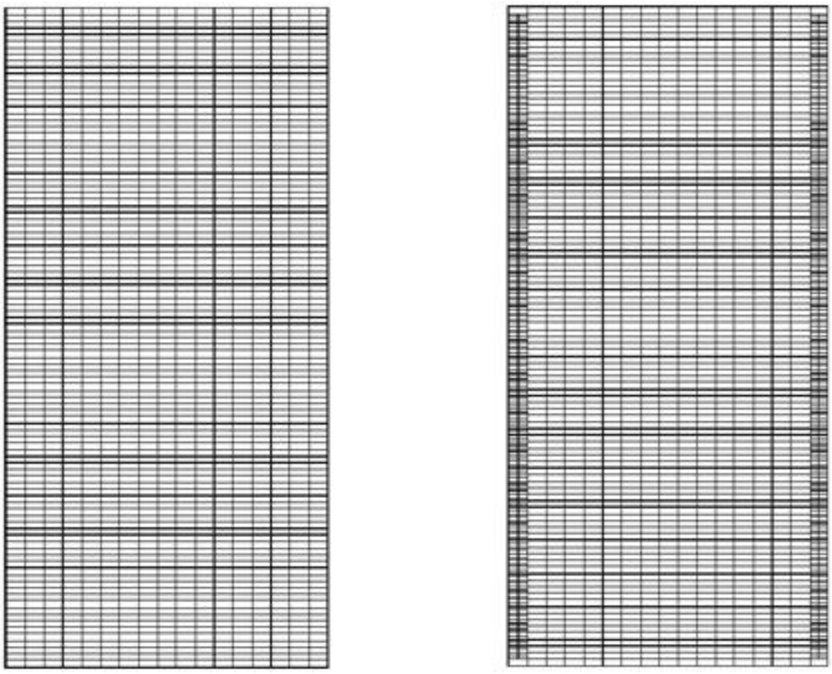
Where, $u^* = \sqrt{\frac{\tau_w}{\rho}}$

y+ depends 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 specific 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 479400 cells was created. 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. 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. But in the present case the method of solution-adaptive refinement was used, where one can add cells where they are needed in the mesh thus enabling the features of the flow field to be better resolved. When adaptation is used properly, the resulting mesh is optimal for the solution because the solution is used to determine where more cells need to be added. Here Y-plus adaptation has been provided to appropriately refine or coarsen the mesh along the wall during the solution process. The minimum allowed y^+ value of 25 and maximum allowed y^+ was taken to refine the mesh near the wall region. After adapting the grid the mesh size increased to 868454 cells. In compressible, 3D steady flow RANS equations implemented to solve the problem. The k-ε, standard k-ω, RSM and Spalart Alamras turbulence models where used and the y^+ values were calculated correspondingly. The maximum and minimum y^+ values corresponding to various turbulent models are shown in Table 1. The grid refinement after the grid adaptation is shown in Fig. 3.

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

Turbulence Model	Y^+(Minimum)	Y^+(Maximum)
K-Epsilon	24.41228	160.2977
K-Omega	24.75391	154.8093
RSM	20.61261	162.1681



(a) Before adaptation
Mesh size = 479400 cells

(b) After adaptation
Mesh Size = 868454 cells

FIGURE 3. Grid layout at the outlet section before and after adaptation

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, Spalart Alamras model and RSM model. It has been observed that with wall y^+ approach for grid refinement process and grid adaptation method 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 duct

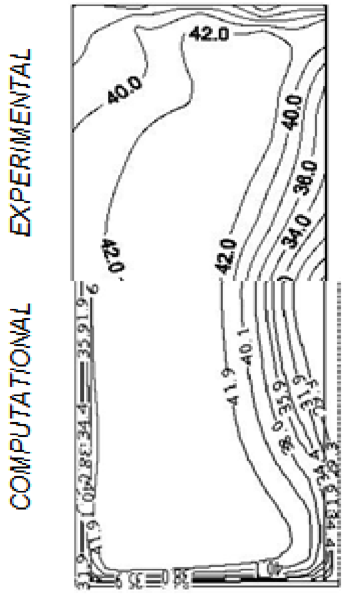


FIGURE 4. Comparison of experimental and numerical results

V. Conclusions

Both experimental and numerical shows that the high momentum fluids shifted towards the outer wall due to the generation of secondary motion and RSM turbulent model predicts the same result with a very good agreement with the experimental result.

References

[1] Rowe, M., 1970. Measurements and computations flow in pipe bends, Journal of Fluid Mechanics, 43(4) 771-783.

[2] Bansod, P and Bradshaw, P., 1972. The Flow in S-shaped Ducts, Aeronautical Quarterly 23 131-140.

[3] Enayet, M. M., Gibson, M. M., Taylor, A. M. K. P. and Yianneskis, M., 1982. Laser Doppler measurements of Laminar and Turbulent Flow in a Bend, Int. Journal of Heat and Fluid Flow 3 211-217.

[4] Lacovides, H., Launder, B. E. and Loizou, P. A., 1987. Numerical Computation of Turbulent Flow through a Squared Sectioned 90° Bend, Int. Journal of Heat and Fluid Flow 8(4) 320-325.

[5] Taylor, A. M. K. P., Whitelaw J. H. and Yianneskis, M., 1982. Curved Ducts with Strong Secondary Motion: Velocity Measurements of Developing Laminar and Turbulent Flow, Trans. ASME, Journal of Fluid Engineering, 104 350-358.

[6] Thangam, S. and Hur, N., 1990. Laminar secondary flows in curved rectangular ducts, Journal of Fluid Mechanics 217 421-440.

[7] Kim, W. J. and Patel, V. C., 1994. Origin and Decay of longitudinal Vortices in the development of flow in a curved rectangular duct, Trans. ASME, Journal of Fluid Engineering 116(3) 45-52.

[8] Gerasimov, A., Modelling Turbulent flow with Fluent, Ansys Inc., Europe, 2006.

[9] Salim, S. M., and Cheah, S. C., Wall Y^+ Stragegy for dealing with wall-bounded Turbulent Flows, Proc. Int. Multiconference of Engg & Comp. Scientists, , Vol. II.

[10] Ariff, M., Salim, S. M., and Cheah, S. C. 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.