

Numerical Validation of Flow Through an S-shaped Diffuser

Nirmal K Das¹, Ashoke K Raman², Amar N Mullick²⁺, B. Halder²,
P. Ray³ and B. Majumdar⁴

¹ Department of Mechanical Engineering, BCET, Durgapur, India

² Department of Mechanical Engineering, NIT Durgapur, India

³ Department of Civil Engineering, NIT Durgapur, India

⁴ Department of Power Engineering, Jadavpur University, India

Abstract. Curved diffusers are mainly used in both open cycle and closed cycle gas turbines. It also uses as air intake of single engine aircraft. The present paper presents the result of an experimental work of mean velocity contours in 2-D form and also the validation of the experimental result with numerical simulations of different turbulence models based on the wall y^+ approach. Different turbulence models available in Fluent code are used to solve the closure problem. Wall y^+ values are considered as a guiding parameter to select the appropriate grid configuration and turbulence model. The geometry of the diffuser is chosen as rectangular cross section with an area ratio of 2.0. The experiment is carried out at $Re=1.56 \times 10^5$ based on the mass averaged velocity and hydraulic mean depth of the inlet section. In numerical analysis wall y^+ value in the laminar sub-layer near the solid wall and at the fully developed layer are evaluated and checked for the values. Based on these criteria it may be concluded that the mesh resolved in the fully turbulent region are sufficiently accurate in terms of the qualitative features of the mesh independency test. On examining the contours at the exit section of all the turbulence models it has been observed that the RSM model predicts the best result as it has got a good qualitative matching with the experimental result.

1. Introduction

Diffuser is a mechanical device usually made in the form of a gradual conical expander intended to convert the dynamic pressure into static pressure of the fluid flowing through it. Diffusers are made in different shapes namely axial, radial, and curved to conform to the constraints imposed by the aspects of design. Subsonic curved diffusers, as air intakes, find wide applications in the field of aircraft design especially in military aircrafts in which the engine is frequently carried in the fuselage and the intake is located in an offset position. The performance of such diffusers, not only in terms of the total pressure delivered but also, and more significantly, in terms of the quality (uniformity, velocity and direction) of flow at the engine face affects the response of the engine. Flow development in S-shaped diffusers is complicated and is influenced by different geometrical parameters like area ratio (AR), aspect ratio (AS), total divergence angle (2θ), angle of turn of the centre line ($\Delta\beta$), inflexion in the curvature, and dynamical parameters like inlet Mach number (Ma), inlet turbulence and specifically the angle of attack when mounted on a combat air craft.

In a curved diffuser, due to the presence of centerline curvature, fluid near the flow axis is acted upon by a larger centrifugal force than the fluid near the walls. This centrifugal pressure difference (transverse pressure gradient) forces the faster moving fluid to move outwards pushing the fluid in the boundary layer at the outer wall around the sides towards the inner wall; thus a significant secondary flow (normal to the primary flow direction) is produced.

⁺ Corresponding author. Tel.: + 919830385342
E-mail address: anmullick@gmail.com or amaranth.mullick@me.nitdgp.ac.in

Probably, Fox & Kline, 1962, [1], were the first to carry out a systematic study of flow regimes in curved diffusers and additional studies of flow regimes in 2-D straight walled diffusers. They developed maps of flow regimes for stalled and unstalled curved diffusers for a range of flow turning angle from 0 to 90°, in steps of 10°, with centre line of the curved diffuser having geometry of a circular arc and a linear area distribution normal to the centre line. McMillan, 1982, [2], studied experimentally the mean flow quantities of low speed turbulent flow through a 40° turn, $L/W_1=3.0$, $AR=1.32$, $AS=1.5$ curved diffuser. He observed that secondary motions existing in the form two contra rotating vortices between parallel walls had a great influence on the flow development of the diffuser. Shimizu *et al.*, 1986, [3,4], in their two papers, experimentally studied the performance and internal flow through twisted S-shaped (coiled), U-shaped, and snake shaped bend diffusers having circular cross-section with area ratios varying from 1.3 to 15.9, divergent angles (3° & 6°), and curvature ratios (3 & 6). They found that the separation generated in the upstream portion of the diffuser was detrimental to the performance of the diffusers and could be overcome by varying the inlet velocity distribution as well as proper selection of an optimum combination of the divergent angle and the radius to curvature ratio. Shimizu *et al.*, 1992, [5], experimentally investigated the influence of aspect ratios ranging from 0.25 to 8 on the performance and generation of secondary flow in sinuous conduits of rectangular cross-section. They observed that the hydraulic losses decrease with increase in aspect ratio, and in case of smallest aspect ratio flow separation occurred in downstream sections resulting in large hydraulic losses. The number of secondary vortex pairs was found to increase with the increase in aspect ratio. Wellborn *et al.*, 1994, [6], conducted tests on diffusing S-ducts of circular cross-section with an inlet centerline Mach number of 0.6, Reynolds number of 2.6×10^6 based on inlet centerline velocity and duct inlet diameter, and an existing thin boundary layer. A stream wise flow separation occurred within the diffuser and the resulting counter rotating vortices degraded both the uniformity and magnitude of the total pressure profile. Dominy and Kirkham, 1996, [7], made extensive analysis of the influence of wakes on the flow through diffusing S-shaped annular duct. Based on the inlet velocity distribution (with wakes and without wakes) they found that the simple axial wakes strongly influence the flow, by affecting not only the hub and casing boundary layers but also more significantly the strong secondary flows, resulting in total pressure distortions. Majumdar *et al.*, 1988, [8], experimentally studied flow through a curved diffuser with high aspect ratio ($AS = 6$), $AR = 2$, $\Delta\beta = 90^\circ$, centerline length 0.6m, and $R_c = 0.382$ m. The shift of stream wise bulk flow towards the outside vertical wall was attributed to the centrifugal effect arising out of the centerline curvature. The wall static pressure increased continuously on both convex and concave walls due to diffusion and a pressure recovery close to 50% was obtained. They also observed that except for a small pocket on the convex wall just at the exit there was no separation in the diffuser. Counter rotating vortices were detected from 30° turn of the diffuser. Majumdar *et al.*, 1997, [9], in their study on a 90°/90° S-shaped diffuser of rectangular cross section of $AR = 2$ and inlet $AS = 6.0$ observed the generation of vortical pair of motion in the first half of the diffuser which after the inflexion changed in sense of rotation. They also observed a small pocket of flow reversal near inflexion plane with a lower overall pressure recovery in comparison to a straight diffuser.

The present paper presents a part of the work which involves the investigation of the flow development through an S-shaped diffuser, characterized by mean velocity and transverse velocity distribution, static pressure and total pressure distribution, coefficients of pressure recovery and pressure loss. However, due to the space restriction only the mean velocity distribution in an S-shaped diffuser at an inlet Reynolds number of 1.56×10^5 corresponding to a mass averaged inlet velocity of 40m/s is presented here.

2. Experimental Facility and Instrumentation

Experiments were conducted in the Aerodynamics Laboratory, National Institute of Technology, Durgapur. The measurements were carried out in an open-circuit wind tunnel as shown in Fig 1. The cross-sectional area of the test diffuser was increased by linearly varying the width from 0.05m at inlet to 0.1m at the exit, over the total centre line length of 0.6m while the height was kept constant at 0.1m. The width was equally distributed normal to the centerline. The curved diffuser was fabricated in four

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 and grid adaptation method RSM model provides the best result among all the four turbulence models. Fig. 3 and Fig. 4 show the experimental and numerical result of the mean velocity contours of the S-diffuser at the exit section. 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. 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.

Table 1

Turbulence Model	Y+(min)	Y+(max)
k-epsilon	4.32	225.82
RSM	4.37	227.35
k-omega	4.43	230.73
RNG k-epsilon	4.13	224.52

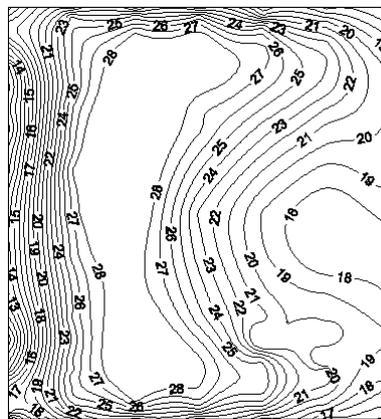
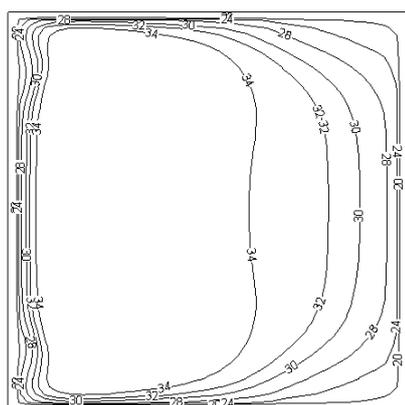
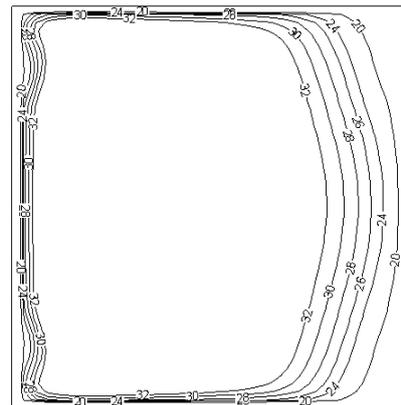


Fig.3 Experimental result of mean velocity contours at the exit section



(a) k- ϵ Model



(b) K- ω Model

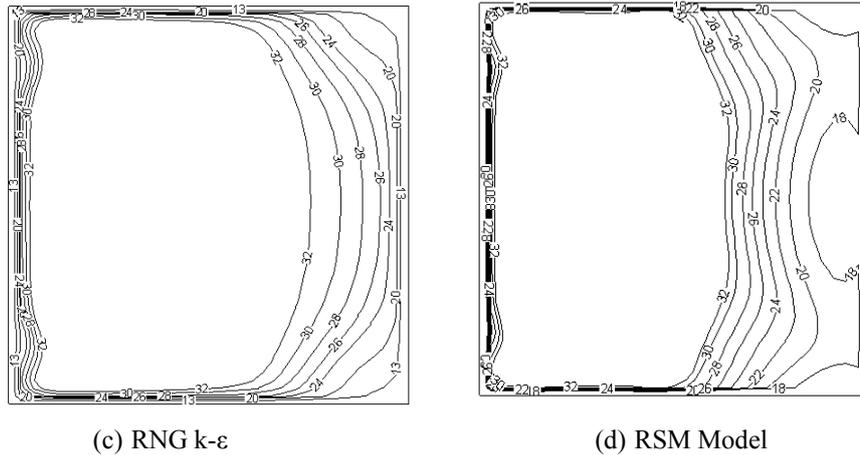


Fig.4 Numerical results of mean velocity contours at the exit section

4. Conclusions

Both experimental and numerical results show that high velocity fluid shifted towards the outer wall due to the generation of secondary motion, which caused due to the combined effect of the centrifugal action of the centerline curvature and adverse pressure gradient and the RSM model with identical inlet conditions with the experimental work predicts the result with a very good matching with the experimental result.

5. References

- [1] R. W. Fox and S. J. Kline. Flow Regimes in Curved Subsonic Diffusers, *Trans ASME, Journal of Basic Engineering*. 1962, 84, pp. 303-316.
- [2] O. J. McMillan. Mean-Flow Measurements of the Flow Field Diffusing Bend, NASA contractor report. 1982, p. 3634.
- [3] Y. Shimizu, M. Nagafusa, K. Sugino and F. Nakamura. Studies on Performance and Internal Flow of Twisted S-Shaped Bend Diffuser-The So-Called Coiled Bend Diffuser: 1st Report, *Trans. ASME Journal of Fluids Engineering*. 1986, 108, pp. 289-296.
- [4] Y. Shimizu, M. Nagafusa, K. Sugino and T. Kubota. Studies on Performance and Internal Flow of U-Shaped and Snake-Shaped Bend Diffusers: 2nd Report. *Trans. of ASME Journal of Fluids Engineering*. 1986, 108, pp. 297-303.
- [5] Y. Shimizu, Y. Futaki and C. S. Martin. Secondary Flow and Hydraulic Losses within Sinuous Conduits of Rectangular Cross section. *Trans. of ASME Journal of Fluids Engineering*. 1992, 114, pp. 593-600.
- [6] S. R. Wellborn, B. A. Reichert and T. H. Okiishi. Study of the Compressible Flow in a Diffusing duct. *Journal of Propulsion and Power*. 1994, 10(5), pp. 668-675.
- [7] R. G. Dominy and D. A. Kirkham. The Influence of Blade Wakes on the Performance of Interturbine Diffusers. *Trans. of ASME, Journal of Turbomachinery*. 1996, 118(2), pp. 347-352.
- [8] B. Majumdar, R. Mohan, S. N. Singh and D. P. Agrawal. Experimental Study of Flow in High Aspect Ratio 90 Deg Curved Diffuser. *Trans. of ASME Journal of Fluids Engineering*. 1988, 120, pp. 83-89.
- [9] B. Majumdar, S. N. Singh and D. P. Agrawal. Flow Characteristics in S-shaped Diffusing Duct. *Int. Journal of Turbo and Heat Engines*. 1997, 14, pp. 45-57.