

Effect of aspect ratio and curvature on characteristics of S-shaped diffusers

Vinit Gupta, Rajneesh Devpura, S.N. Singh* & V Seshadri

Applied Mechanics Department, Indian Institute of Technology, New Delhi 110 016, India

Received 7 June 2000; accepted 26 February 2001

Performance characteristics of S-shaped rectangular diffusers have been investigated using CFD. The characteristics have been obtained for 15°/15°, 22.5°/22.5°, 30°/30°, 45°/45° and 90°/90° diffusers for constant circular center line length of 600 mm having an aspect ratio of 2, 4 and 6 at inlet. It is observed that as the curvature increases, the uniformity of flow at the outlet reduces, the cross flow velocities at the outlet increase and the coefficient of pressure recovery decreases. The effect of aspect ratio on the performance characteristics of these diffusers is also similar, i.e. reduction in aspect ratio reduces flow uniformity and pressure recovery.

Diffusers are used for converting kinetic energy of the fluid to pressure energy. There is a continuous reduction in the flow velocity along the length of the diffuser. Diffusers can be broadly classified into two categories: Straight and Curved Diffusers. The S-shaped diffusers fall in the category of curved diffusers. S-shaped diffusers are mainly used in aerospace applications as intake ducts.

The flow characteristics at the outlet of S-shaped diffusers play a crucial role in the design of the downstream elements, namely, the compressor, combustor, etc. The flow in these diffusers becomes complicated due to the presence of inflection point in curvature in the direction of flow resulting in strong pressure driven stream-wise vortices. The flow characteristics are also affected by the inlet conditions and the aspect ratio of the diffuser. The flow displays three-dimensional features, mainly due to curvature induced stream-wise vorticity and secondary motion associated with diffusers.

Studies have been conducted on constant area S-shaped ducts with circular cross-section¹⁻³. The studies have shown the presence of a pair of counter rotating vortices within the duct. Taylor *et al.*^{4,5} have reported measurements of flow in circular and square cross-section S-shaped ducts for developing laminar and turbulent flow. These measurements show a higher degree of secondary motion for laminar flow at the outlet due to the thicker boundary layer at inlet. Rojas *et al.*⁶ investigated the effect of developing laminar and turbulent flow on the characteristics of C and S-shaped diffusers (22.5°/22.5°), having moderate

curvature. The area ratio and aspect ratio at the exit taken in both the cases were 1.5 and 0.67, respectively. The development of secondary motion was observed to be similar to that observed in constant area ducts (Taylor *et al.*⁴), with the magnitude of cross flow velocity, reducing towards the exit of the duct. The pressure recovery achieved was 1/5th of the velocity head. Guo and Seddon^{7,8} studied the effect of swirl in S-shaped diffusing duct used in modern aircraft. Different techniques have been proposed to improve the performance of the diffuser. Seddon⁹ has explained the phenomenon of self-generated swirl and suggested means to overcome them. Shimzu *et al.*^{10,11} investigated the effect of inlet boundary layer thickness on the performance of these diffusers. Whitelaw and Yu¹² have studied the turbulent flow characteristics in an S-shaped diffuser. They observed attenuation of counter rotating vortices in the second half, when a skewed velocity profile was fed at the inlet. Lien and Leschinger¹³ have demonstrated that the mean flow features of diffusers and in curved ducts can be well predicted and are insensitive to the turbulence model used, on the contrary turbulence field energy is very sensitive to the variation of the turbulence model. Majumdar *et al.*¹⁴ have characterised the flow in S-shaped diffuser, with an area ratio of 2. The results obtained by their experiment have been used to validate the CFD code, used in the present study. A survey of the available literature reveals that many experimental investigations have been reported to understand the flow complexities in S-shaped diffusers but no systematic attempt has been made to unravel the flow features in S-shaped diffusers with different area ratios and curvatures, either

*For correspondence

experimentally or using Computational Fluid Dynamics (CFD).

The results presented in the present paper have been obtained using commercial CFD software FLUENT¹⁵. CFD has distinct advantages over experimental techniques, like high speed and low cost with reasonable accuracy. It can also be used to simulate flow in complex geometries with immense ease.

Mathematical Formulations

The governing equations are:

$$\rho \frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i) = S_m \quad \dots (1)$$

$$\rho \frac{Du_i}{Dt} = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial u_l}{\partial x_l} \right) \right] + \frac{\partial}{\partial x_j} (-\overline{\rho u_i' u_j'}) \quad \dots (2)$$

Where, ρ is the density of the medium (air, in the present case); u_i is the mean velocity in the three directions for three values of $i = 1, 2, 3$, respectively; μ is the viscosity of the medium, P is the static pressure and u_i' is the perturbation in velocity due to turbulence.

Additional terms $-\overline{\rho u_i' u_j'}$, in the momentum equations, are the Reynolds stresses and these must be modeled in order to have the close form solution. Various turbulence models have been developed to incorporate the Reynolds stresses. The k - ϵ turbulence model has been used in the present study. The selection of the k - ϵ turbulence model has been made on the basis of the experience of Koutmos and McGuirk¹⁶ and Majumdar¹⁷. They have used standard k - ϵ turbulence model for predictions in a dump diffuser and S-shaped diffusers, respectively. In S-shaped diffusers, Majumdar¹⁷ has observed no separation zone and has reported that k - ϵ model works reasonably well for all engineering purposes. Boussinesq hypothesis is used to relate the Reynolds stresses to the mean velocity gradient as described in the form of Eq. (3);

$$-\overline{\rho u_i' u_j'} = \mu_t \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \left(\rho k + \mu_t \frac{\partial u_l}{\partial x_l} \right) \delta_{ij} \quad \dots (3)$$

where, μ_t is the eddy viscosity and k is the turbulence kinetic energy.

Two additional transport equations one for turbulent kinetic energy (k) and another for the turbulence dissipation rate (ϵ) are solved to evaluate μ_t , which is computed as,

$$\mu_t = \rho C_\mu k^2 / \epsilon \quad \dots (4)$$

Where, C_μ is a constant.

The additional equations are

$$\rho \frac{D\epsilon}{Dt} = \frac{\partial}{\partial x_i} \left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial x_i} \right] + G_\epsilon + G_b - \rho \epsilon - Y_m \quad \dots (5)$$

$$\rho \frac{Dk}{Dt} = \frac{\partial}{\partial x_i} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_i} \right] + C_{1\epsilon} \frac{\epsilon}{k} (G_k + C_{3\epsilon} G_b) - C_{2\epsilon} \frac{\rho \epsilon^2}{k} \quad \dots (6)$$

Where, G_k is the generation of turbulent kinetic energy due to the mean velocity gradient.

$$G_k = \mu_t S^2 \quad \dots (7)$$

Here S is the modulus of the mean rate of stress tensor, defined as

$$S = (2 S_{ij} S_{ij})^{1/2} \quad \dots (8)$$

S_{ij} is given by,

$$S_{ij} = \frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \quad \dots (9)$$

G_b is the generation of turbulent kinetic energy due to buoyancy. In present study, it has been taken as zero because of zero temperature gradient. The term Y_m represents the contribution of the fluctuating dilation in compressible turbulence to the overall dissipation rate and is taken as zero, since the flow considered is incompressible. $C_{1\epsilon}$ and $C_{2\epsilon}$ are constants. σ_k and σ_ϵ are the turbulent Prandtl numbers for k and ϵ , respectively.

The value of constants in the present study taken for the turbulence model are the standard values

reported in literature and are: $C_{2\varepsilon} = 1.92$; $C_{1\varepsilon} = 1.44$; $C_\mu = 0.09$; $\sigma_k = 1.0$; and, $\sigma_\varepsilon = 1.3$

These values have been found to work fairly well for a wide range of wall bounded and free shear flows.

Validation of the code

The geometry, used for validation, is same as that used by Majumdar *et al.*¹⁴ The geometry of the diffuser has been shown in Fig. 1. Velocity profiles have been obtained at various stations, which are nearly similar to those used in the experiments¹⁴ (Fig. 2). The coefficient of pressure recovery value predicted (41.7%) agrees reasonably well with the measured values (45.6%). The discrepancies observed in the predicted and the experimental results could be due to error in the simulation of the inlet velocity profile as that of Majumdar¹⁴ and the limitations of the turbulence model to handle zones of steep velocity gradient. Modifications in the inlet velocity profile showed improvement in matching but the results of validation have been given only for that inlet velocity profile which in view of the authors was true representative of the velocity profile measured by Mazumdar *et al.*¹⁴

Investigation of geometries

To establish the effect of curvature, five combinations of angles of turn, namely, $15^\circ/15^\circ$, $22.5^\circ/22.5^\circ$, $30^\circ/30^\circ$, $45^\circ/45^\circ$ and $90^\circ/90^\circ$ for an S-shaped rectangular diffuser, have been selected. The other parameters, which were kept constant, are aspect ratio at inlet (6), area ratio (2), centerline length (600 mm) and centerline shape (circular).

For studying the effect of aspect ratio, aspect ratios of 2, 4 and 6 at inlet were chosen for $30^\circ/30^\circ$ and $90^\circ/90^\circ$ S-shaped diffusers. The other parameters were kept constant as above.

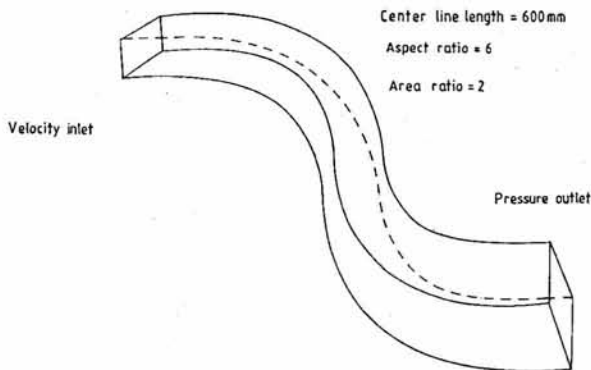


Fig. 1 — Geometry of S-shaped diffuser used for validation

Specification of boundary conditions

A uniform velocity profile is specified at the inlet, with magnitude equal to 40m/s. Atmosphere pressure is specified at the exit as an outlet boundary condition. Turbulence intensity is specified as 3-4% at the inlet and 7-8% at the outlet, for purpose of initialization to have faster convergence.

Results and Discussion

To clearly bring out the effect of radius of curvature and aspect ratio, results in the subsequent discussion are divided into two parts. The analysis is restricted to the pressure recovery and outlet velocity profile and only selected graphs are presented here.

Effect of variation of radius of curvature on velocity profile at the outlet

Velocity has been broken up into two components for studying the behaviour of the flow at the outlet. The components, which have been chosen, are longitudinal velocity and the in-plane velocity. The profiles of both the components have been carefully observed at the outlet. Isovelocity contours are presented to depict the variation of longitudinal velocity at the outlet, whereas in-plane velocity

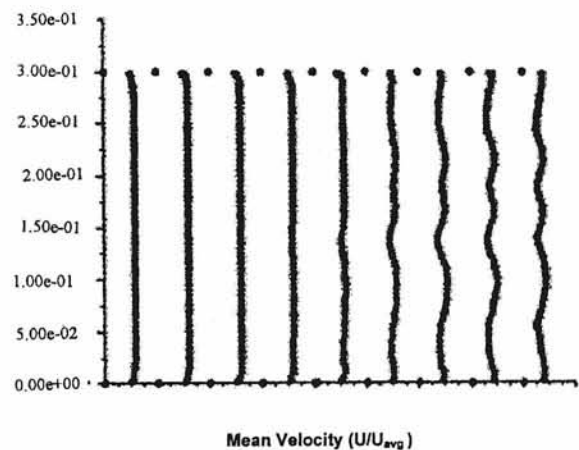


Fig. 2a — Mean velocity at outlet at various locations using CFD

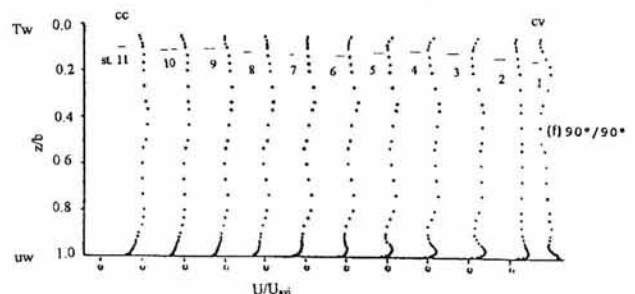


Fig. 2b — Mean velocity at various locations at the outlet (experimentation)

vectors have been drawn for depicting the variation of cross flow velocity at the outlet in Figs 3 - 5.

For $15^\circ/15^\circ$ diffuser, longitudinal velocity is observed to be uniform in the core and it is seen to decrease with a steep gradient to zero at the wall, as expected due to the wall effect. The in-plane velocity vectors depicting cross flow show the presence of one pair of counter rotating vortices, though of very low intensity, which are only prominent at the left and right boundaries of the outlet plane and negligible along the concave and convex walls. The maximum value of the cross flow velocity is observed to be 0.8 m/s. For $22.5^\circ/22.5^\circ$ diffuser (Fig. 3a, b), longitudinal velocity is again observed to be uniform in the core, and the profile is almost the same as that observed for $15^\circ/15^\circ$ diffuser. The cross flow velocity vectors again show a pair of counter rotating vortices, with a slightly greater intensity. The maximum cross flow velocity is observed to be 0.9 m/s. For $30^\circ/30^\circ$ diffuser, a similar trend as that in $22.5^\circ/22.5^\circ$ diffuser is observed, with intensity of vortices increasing a bit more. For $45^\circ/45^\circ$ diffuser (Fig. 4a,b), uniformity of

the longitudinal velocity is lost and it is observed to vary along the width, with the highest value in right and left top halves. The in-plane velocity vectors still show a similar sort of configuration with further increase in the intensity of the counter rotating vortices, the maximum being 1.8 m/s. For the $90^\circ/90^\circ$ diffuser (Fig. 5a,b), flow is observed to be highly non-uniform at the outlet, showing no particular behavior. There are three pairs of vortices observed at the outlet, instead of the one pair of counter rotating vortices, observed in previous cases. Mazumdar *et al.*¹⁴ have also reported presence of about three pair of vortices at the outlet of $90^\circ/90^\circ$ diffuser.

Effect of variation of radius of curvature on pressure recovery and cross flow velocity

Fig. 6a gives the variation of average cross flow velocity at outlet. It is seen that the magnitude of cross flow velocity increases with increase in curvature. Fig. 6b gives the variation of pressure recovery coefficient as a function of angle of turn. It

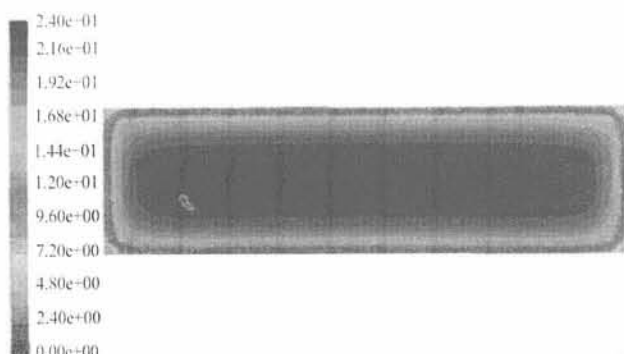


Fig. 3a — Contours of longitudinal velocity at outlet for $22.5^\circ/22.5^\circ$ diffuser

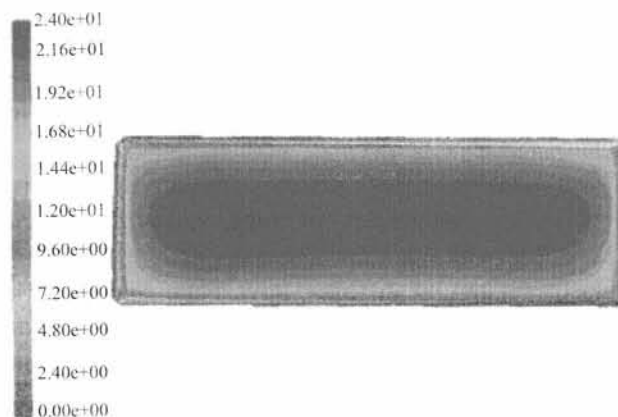


Fig. 4a — Contours of longitudinal velocity at outlet for $45^\circ/45^\circ$ diffuser



Fig. 3b — Velocity vectors showing cross flow velocity at outlet for $22.5^\circ/22.5^\circ$ diffuser

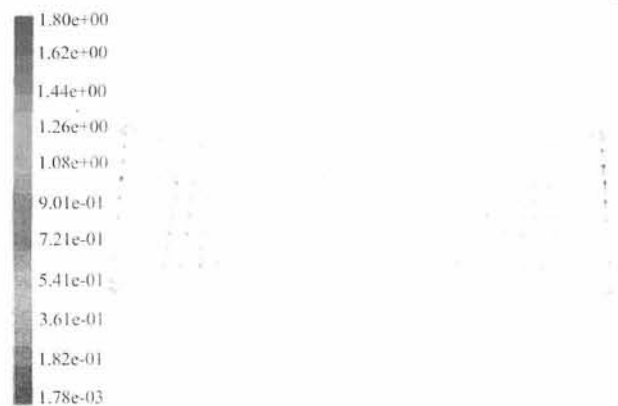


Fig. 4b — Velocity vectors showing cross flow velocity at outlet for $45^\circ/45^\circ$ diffuser

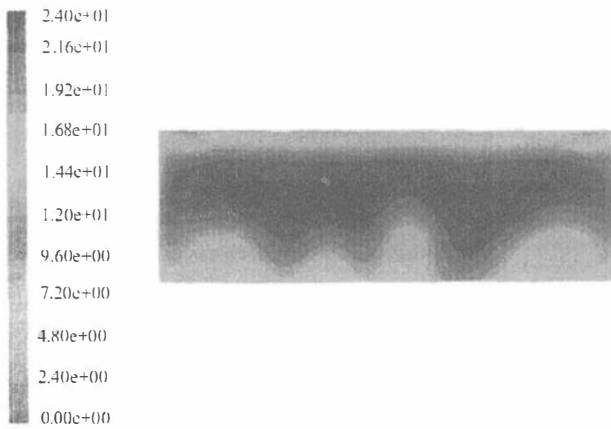


Fig. 5a — Contours of longitudinal velocity at outlet for 90°/90° diffuser

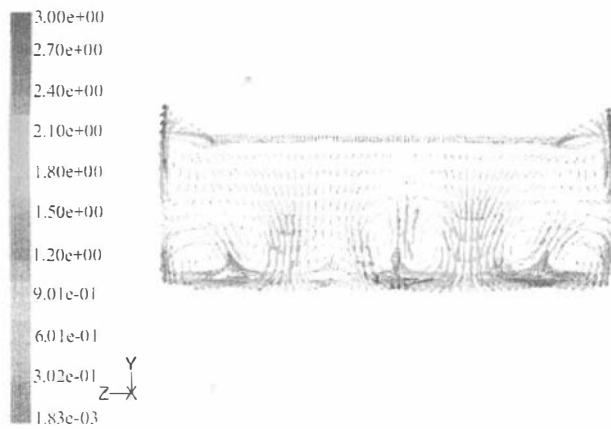


Fig. 5b — Velocity vectors showing cross flow velocity at outlet for 90°/90° diffuser

is observed, that the coefficient of pressure recovery decreases, as the angle of turn of the diffuser increases. The pressure recovery coefficient for 15°/15° combination is predicted as 65.1% which reduces to 62.5% for 45°/45° "S" diffuser, a fall of about 4%. For 90°/90° angle of turn, the pressure recovery falls to 41.7% showing a fall of approximately 30%.

It can be seen from the predicted trends that as the curvature of the diffuser increases, longitudinal velocity profile at the outlet becomes less uniform. This non-uniformity at outlet can be attributed to increase in intensity of secondary flows, which are generated due to centrifugal forces, and are more predominant in the case of high curvature diffusers. As a result of this, high curvature diffusers have less uniform flow at the outlet, when compared with low curvature diffusers. The generation of the counter rotating vortices can be attributed to the centrifugal

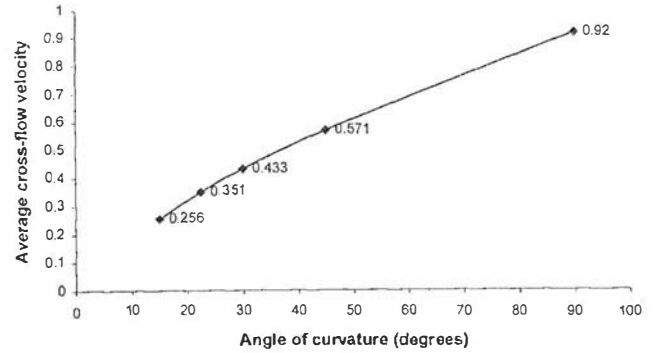


Fig. 6a — Variation of average cross-flow velocity at outlet with angle of turn

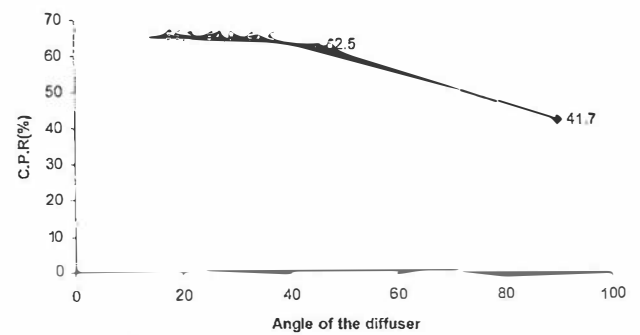


Fig. 6b — Coefficient of pressure recovery for various angles of turn

forces, which try to push the fluid towards the concave wall, but the fluid turns due to the normal pressure from the concave wall, leading to the generation of the vortices. The increasing intensity of the counter rotating vortices with increase in curvature also suggests the involvement of centrifugal forces, generated due to the curvature.

Effect of variation of aspect ratio on velocity profile at the outlet

Figs 7 and 8 show the variation of longitudinal velocity and cross flow velocity at the outlet for 30°/30° diffuser having an aspect ratio of 2 and 6 at the inlet, respectively. It is observed that as the aspect ratio increases, flow gains uniformity at the outlet for the 30°/30° diffuser. As a result, more uniform flow is observed at the outlet for an aspect ratio of 6 than that observed for an aspect ratio of 2. It may be due to the slow penetration of corner effects to the core of the flow for the case of high aspect ratio diffuser. A high value of cross flow velocity is observed near the parallel walls and very low magnitude is observed in the center of the outlet plane (Fig.8b). For 90°/90°

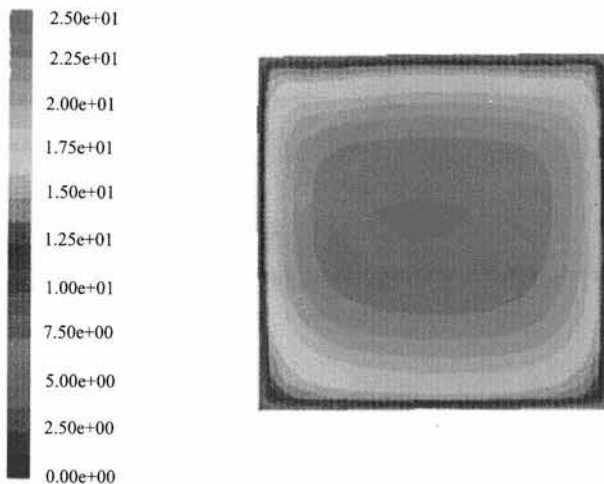


Fig. 7a — Contours of longitudinal velocity at outlet for 30°/30° diffuser having an aspect ratio of 2 at inlet

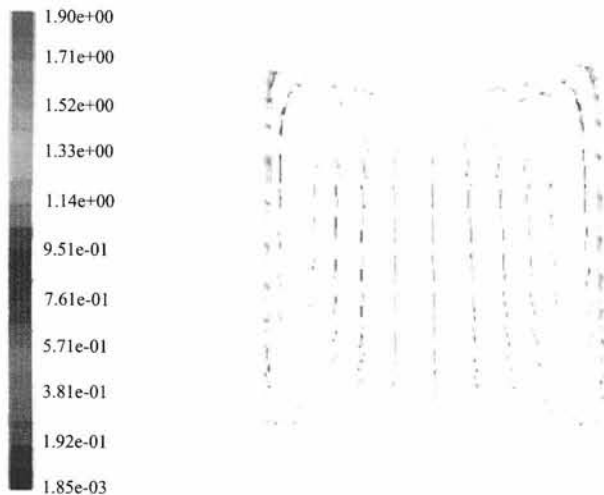


Fig. 7b — Velocity vector showing cross flow velocity at the outlet for 30°/30° diffuser having an aspect ratio of 2 at inlet

diffuser the flow is accompanied with high level of secondary motion, making the secondary flow more prominent in the case of aspect ratio 6 than that for aspect ratio 2, which can also be seen in the respective cross velocity profiles (Figs. 5b and 9b). For 90°/90° S-shaped diffuser, the pair of contra rotating vortices for two aspect ratio shows a tendency of breaking into larger number of pair of contra rotating vortices (Fig. 9b).

Effect of variation of aspect ratio on Coefficient of pressure recovery and cross flow velocity

The variation of coefficient of pressure recovery with aspect ratio for 30°/30° and 90°/90° diffusers are shown in Figs. 10a and 10b, respectively. For 30°/30°

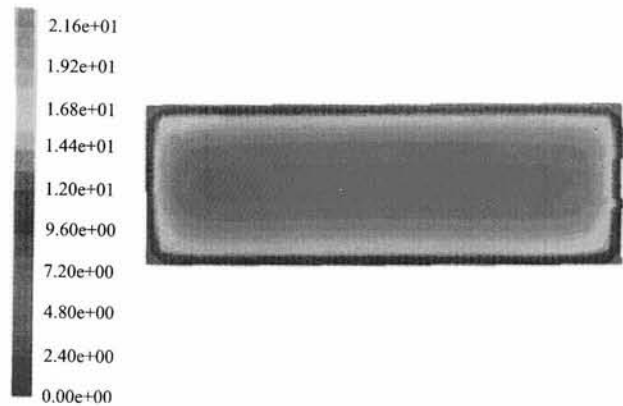


Fig. 8a — Contours of longitudinal velocity at outlet for 30°/30° diffuser having an aspect ratio of 6 at inlet

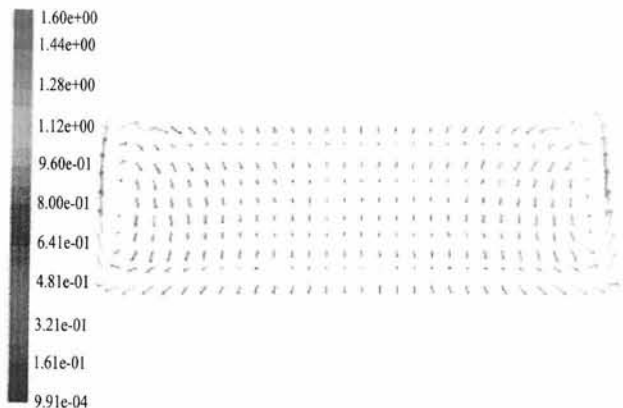


Fig. 8b — Velocity vectors showing cross flow velocity at outlet for 30°/30° diffuser having an aspect ratio of 6 at inlet

diffuser the pressure recovery coefficient increases with increase in aspect ratio, the increase is of the order of around 15% for aspect ratio of 6.0 (Fig. 10a). This can be attributed to the increase in effective diffusion area for high aspect ratio diffusers. For 90°/90° diffuser, the flow has high degree of secondary motion, which increases with aspect ratio, therefore the loss of energy to secondary motion is far greater than the gain due to increase in effective diffusion area. Hence, an overall decrease in CPR is observed with increase in aspect ratio for 90°/90° diffuser (Fig. 10b). The fall in the pressure recovery for aspect ratio 6.0 is the order of 20% in comparison to aspect ratio 2.0.

Conclusions

The flow field at outlet in all the rectangular cross section diffusers studied are observed to contain a pair of counter rotating vortices and the intensity of these vortices increases with increase in angle of turn of the

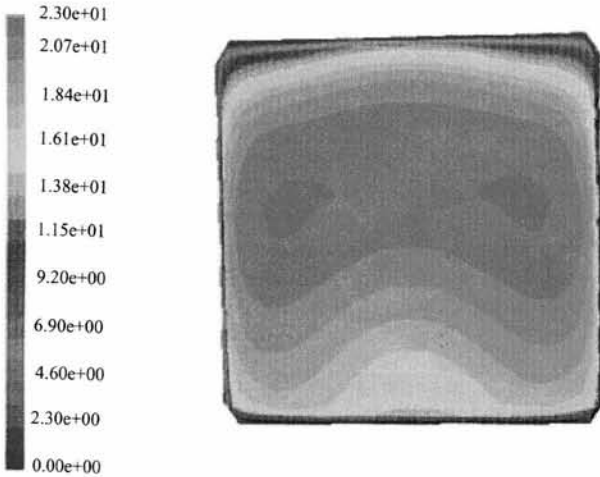


Fig. 9a — Contours of longitudinal velocity at outlet for 90°/90° diffuser having an aspect ratio of 2 at inlet

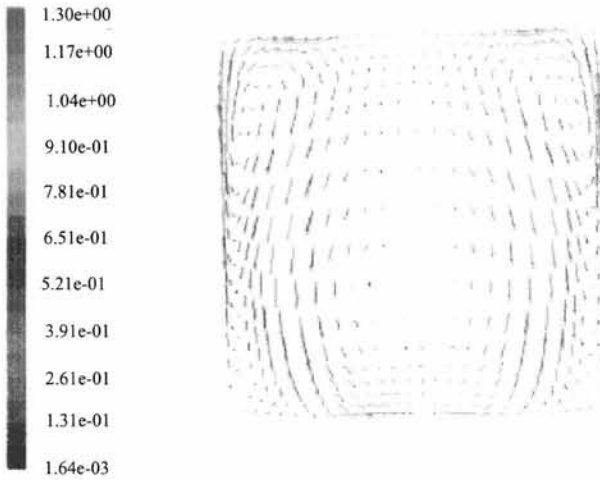


Fig. 9b — Velocity vectors showing cross flow velocity at outlet for 90°/90° diffuser having an aspect ratio of 2

diffuser. The increase in the intensity of the vortices can be attributed to the centrifugal forces, whose dominance increases with the curvature of the diffuser. Also, the uniformity of longitudinal velocity has been observed to decrease with the angle of turn of the diffuser, whereas uniformity of flow is seen to increase with aspect ratio, since the influence of the corner effect reduces.

The coefficient of pressure recovery decreases with increase in the curvature of the diffuser, whereas the magnitude of cross flow velocity increases with the curvature, leading to a lower pressure recovery for the high curvature diffusers. A higher value of coefficient of pressure recovery is observed for high value of aspect ratio diffusers for low angle of turn, whereas the inverse is observed for high angle of turn.

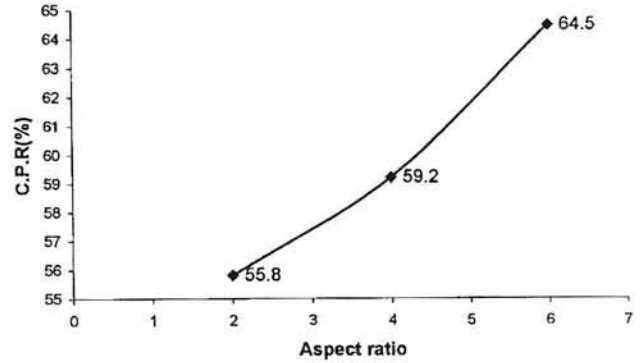


Fig. 10a — Variation of coefficient of pressure recovery with aspect ratio for 30°/30° diffuser

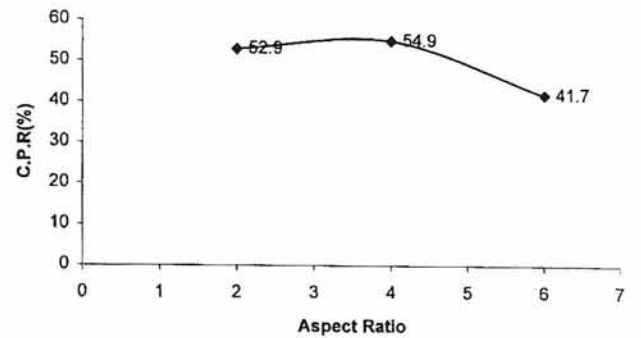


Fig. 10b — Variation of coefficient of pressure recovery with aspect ratio for 90°/90° diffuser

On the basis of this study, it can be concluded that the desired S-shaped diffuser would be one with low curvature and high aspect ratio. The angle of turn should not exceed 30°/30° combination if reasonable flow uniformity is required at the outlet without sacrificing the pressure recovery.

Nomenclature

k	Turbulent kinetic energy
$U_{\max}$	Maximum velocity
U	Mean longitudinal velocity
ϵ	Turbulence dissipation rate
ρ	Density
μ	Dynamic viscosity
ν	Kinematic viscosity
μ_t	Eddy/turbulence viscosity
A_f	Area of face
V	Cell volume
V_f	Mass flux (velocity) through the faces
α	Under relaxation factor
S_m	Mass added to continuous phase
i, j	Indices of tensorial notation
x	Longitudinal coordinate
r	Radial coordinate
p	Static pressure, Pa
$C_{1\epsilon}, C_{2\epsilon}, C_\mu, \sigma_k, \sigma_\epsilon$	Constants of the turbulence model

G_k	Generation term (kinetic energy)
G_b	Generation term (buoyancy)
Y_m	Fluctuating dilation

References

- 1 Rowe M, *J Fluid Mech*, 43 (1970) 771-783.
- 2 Bansod P & Bradshaw P, *Aeronaut Q*, 23 (1972) 141-145.
- 3 Butz L A, *Turbulent flow in S-shaped ducts*, M.Sc. Thesis, Purdue University, USA, 1979.
- 4 Taylor A M P K, Whitelaw J H & Yianneskis M, *Developing Flows in S-shaped Ducts. Part I – Square Cross-section*, Report No. FS/81/22, Mechanical Engineering Department, Imperial College of Science, Technology and Medicine, London, UK, 1981.
- 5 Taylor A M P K, Whitelaw J H & Yianneskis M, *Developing Flows in S-shaped Ducts. Part II – Circular Cross-section*, NASA Contract Report 3759, 1984.
- 6 Rojas J, Whitelaw J H & Yianneskis M, *Flow in Sigmoid Diffusers of Moderate Curvature*, Report No. FS/83/211/22, Mechanical Engineering Department, Imperial College of Science, Technology and Medicine, London, UK, 1983.
- 7 Guo R W & Seddon J, *Aeronaut Q*, (1983) 99-129.
- 8 Guo R W & Seddon J, *Aeronaut Q*, (1983) 130-146.
- 9 Seddon J, *Aeronaut J*, (1984) 117-127.
- 10 Shimizu Y, Nagafusa M, Sugino K & Nakamura F, *Trans ASME, J Fluids Eng*, 108 (1986) 289-296.
- 11 Shimizu Y, Nagafusa M, Sugino K & Kubota T, *Trans ASME, J Fluids Eng*, 108 (1986) 297-303.
- 12 Whitelaw J H & Yu S C M, *Flow Measure Instrum*, 4 (1993) 171-179.
- 13 Lien F S & Leschziner M A, *Computational modeling of 3D turbulent flow in S-diffuser and transition ducts*, in *Engineering turbulence modeling and experiments – 2*, edited by Rodi W & Martelli F (Elsevier Science Publishers), 1993, 217.
- 14 Majumdar B, Singh S N & Agarwal D P, *Int J Turbo Jet Engines*, 14 (1997) 45-57.
- 15 *Fluent 5 User's Guide* (Fluent Incorporation, USA), 1998.
- 16 Koutmos P & McGuirk J J, *J Mech Eng Sci*, 23 (1989) 319-331.
- 17 Majumdar B, *Flow investigations in curved diffusers*, Ph.D. Thesis, Indian Institute of Technology, Delhi, 1994.