Characteristics of 90°/90° S-Shaped Diffusing Duct using SST k-ω Turbulent Model

Manideep Roy[1]

[1]*B.Tech, Department of Mechanical Engineering, National Institute of Technology Durgapur, West Bengal, India*

***

Abstract - *Characteristics of S-shaped diffusing ducts are explored using CFD. The characteristics are achieved for turning angle* $90^0/90^0$ *with constant circular centerline length 600 mm, constant outlet cross-sectional area 100x300* mm^2 *and constant inlet cross-sectional area 50x300*mm^2*. Incompressible flow analysis has been done using SST k-ω turbulent model. Flow separation is observed near the top and the bottom wall of the diffuser. Cross-flow velocity vector shows the formation of a counter rotating vortex pair at the exit plane. The variations in the total pressure loss coefficient and static pressure recovery coefficient and are obtained for different x/c value.*

Key Words: CFD, flow separation, S diffusers, SST k-ω

1. INTRODUCTION

Diffusers are utilized in many systems to slow down flow for the conversion of dynamic pressure into static pressure. Based on the application, the diffusers are produced in many sizes and shapes. S-shaped diffusing ducts are used as interconnectors between gas turbine engine components and as intake manifold for aircraft engines.

The purpose of the diffuser design is to calculate pressure recovery, losses, and flow uniformity at the outlet.

Characteristics of flow in the diffusers are strongly dependent on the ratio of the outlet to the inlet area, length of the centerline, the throat width ratio, the velocity at the inlet, the turbulence intensity profile and the angle of turn.

Strong secondary motion occurs due to disparity between radial pressure gradient and centrifugal force resulting in uneven flow distribution and increased losses.

Nur Hazirah et al. [5] investigated pressure recovery and flow uniformity of turning diffusers using baffles. Velocity at the outlet is found to be more uniform with the use of baffles.

Saha et al. [7] conducted flow simulation in 22.5°/22.5° curved diffuser having a constant circular outlet and changing cross-section at inlet such as elliptic, semicircular, rectangular, square, thereby elliptical shape has been obtained as the optimized one.

Gupta et al. [1] performed flow simulation in 15°/15°, 22.5°/22.5°, 30°/30°, 45°/45°, 90°/90° at a constant circular centerline length of 600 mm. with an aspect ratio of 2, 4, 6 at the inlet. It has been observed that as the curvature and aspect ratio increases, the flow uniformity at the exit reduces, the in-plane velocity increases and the pressure recovery coefficient decreases.

Sullery et al. [8] performed a comparison of performance for curved and straight diffuser. It has been found that for straight diffusers, pressure recovery is better than for curved diffuser and that flow turbulence has a greater influence on the pressure recovery of bent diffuser.

Paul A R et al. [6] performed comparative studies on flow control using vortex generators immersed in rectangular S-duct diffusers. Flow separation was found only for bare S-duct whereas no flow separation was found in S duct with the immersed vortex generator

B.Majumdar et al. [4] carried out experimental measurements to investigate characteristics of flow in the S-shaped diffusing duct with area ratio 2.0 and aspect ratio 6.0 at the inlet. It was found that the total pressure recovery is less *i.e.* 46 percent of the inlet dynamic pressure. The reversal of the flow in the inflection plane and the uniformity of the flow at the outlet were also observed.

Jihyeong Lee et al. [3] studied the effect of boundary layer suction on flow distortion at the inlet using subsonic S diffuser. It was found that the flow separation was improved by suction of the boundary layer.

M.A.A Halim et al. [2] investigated the development of k-ω and k-ε turbulent models in modelling the flow and performance of S-shaped diffusers. It was found that k-ω turbulent model gives better results than the k-ε model.

Thenambika V et al. [9] performed a numerical study on recovery of static pressure and loss of total pressure using a vortex generator at Mach no 0.6 and 1.0 thereby, obtaining better flow values for Mach no 0.6 than Mach no 1.0.

The purpose of this study is to analyze the performance characteristics of 90°/90° S- shaped diffuser using SST k-ω turbulent model.

2. MATHEMATICAL FORMULATION

Any fluid domain is divided into infinitesimally small control volumes where the equations of fluid flow are written in the form of partial differential equations. The continuity equation for incompressible and steady flow (density and time independent) is as follows:

$$\frac{\partial \overline{u}_i}{\partial x_i} = 0$$

The RANS (Reynolds Average Navier Stokes) equations used for the present turbulent models are:

$$\frac{\partial \overline{u}_i}{\partial t} + \overline{u}_j \frac{\partial \overline{u}_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_i} + \upsilon \frac{\partial^2 \overline{u}_i}{\partial x_j \partial x_j} + \frac{\partial(-\overline{u_i' u_j'})}{\partial x_j}$$

where $-\overline{u_i' u_j'}$ is called the Reynolds stress which is approximated by Boussinesq Hypothesis as given below:

$$-\overline{u_i' u_j'} = \nu_t \left(\frac{\partial \overline{u}_i}{\partial x_j} + \frac{\partial \overline{u}_j}{\partial x_i} \right) - \frac{2}{3} k \delta_{ij}$$

3. GEOMETRY USED

The performance parameters of rectangular S-shaped diffuser is studied using SST K-ω turbulent model. The geometrical parameters of the diffuser (as shown in Figure 1. is same as that of Majumdar et al [4].)

1) Cross-sectional area of inlet = 50x300 mm^2
2) Cross-sectional area of outlet = 100x300 mm^2
3) Centreline length = 600 mm.
4) The angle of turn = 90°/90°

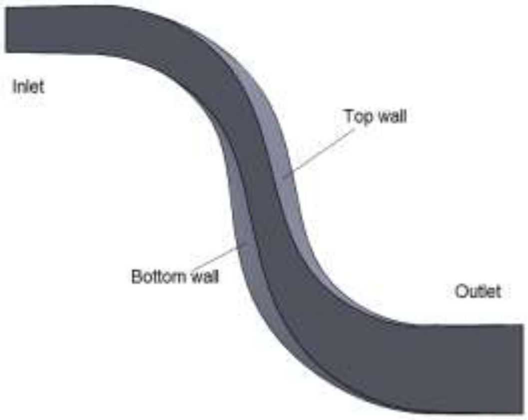

Fig-1. S-shaped diffusing duct used in the present study.

4. CODE VALIDATION

The geometry used for the purpose of validation is similar to that of Majumdar et al. [4]. The experimental result showed that the overall Cp value is 46% of the inlet dynamic pressure. A numerical investigation is done using SST k-ω turbulent model. The Cp value obtained is 51.32% of the inlet dynamic pressure. The difference in Cp value observed in the numerical and experimental results may be due to an error in the input velocity determination as that of Majumdar [4]. The error can also be caused by the density and viscosity of air, which are temperature dependent. Change in the inlet speed showed an improvement in the results.

5. SOLUTION SCHEME

Second-order upwind discretization scheme is applied to each equation for greater accuracy. Pressure-velocity coupling is performed using the SIMPLE algorithm. The residuals were continuously computed for y-velocity, x velocity, continuity, z-velocity, ω and, k. The convergence criteria for the residues were set to 10^{-5}.

6. BOUNDARY CONDITIONS

The velocity at the inlet is taken as 40 m/sec. Atmospheric pressure was specified at the outlet. The turbulent intensity is taken as 4% at the inlet and 8% at the outlet for the purpose of initialization to have faster convergence. No-slip boundary condition is specified at all walls.

7. RESULTS AND DISCUSSION

Using SST k-ω turbulent models, velocity variation at the longitudinal mid plane, cross flow velocity vectors at the exit plane, wall shear stress and different parameters of performance such as coefficient of static pressure recovery and coefficient of total pressure loss have been investigated.

7.1 Velocity variation at the longitudinal mid-plane and cross flow velocity vector at the exit:

Figure 2. illustrates the velocity variation at the longitudinal midplane of the diffuser. Zero or negative values of velocity indicates flow separation. Flow separation is observed near the top and the bottom wall.

Figure 3. illustrates the evolution of a pair of vortices rotating in the opposite direction due to centrifugal forces. The centrifugal forces try to push the fluid away from the top wall, but the fluid turns due to normal pressure from the bottom wall, leading to the generation of vortices.

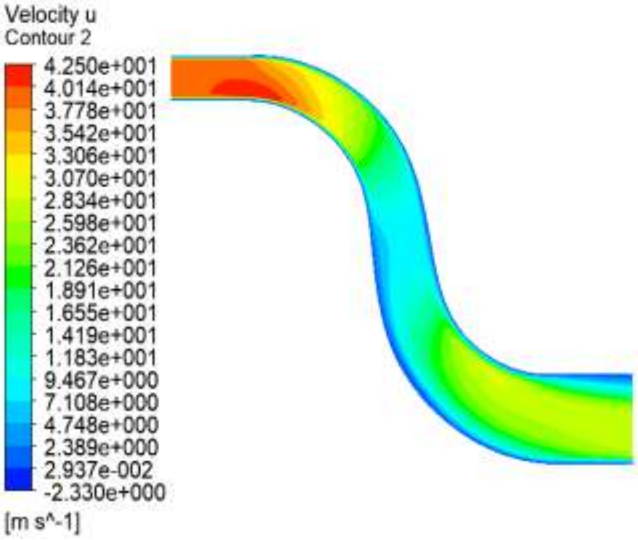

Fig-2. Velocity contour at the longitudinal mid plane of the diffuser.

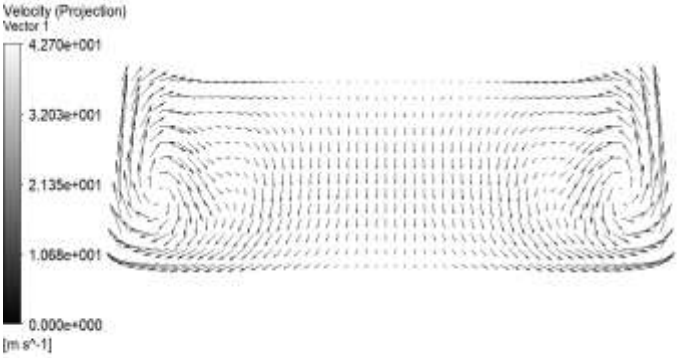

Fig.-3. Crossflow velocity vector at the exit

7.2 Effect of variation of x/c on static pressure recovery coefficient:

Figure 4. illustrates that the coefficient of static pressure recovery (Cp) first increases, then decreases slightly (at point 4) after which it increases to the exit plane. It is obvious from the general trend that the Cp value will increase with the increase in x/c. Here, the decrease in Cp is found because of the flow separation occurring at the bottom and the top walls.

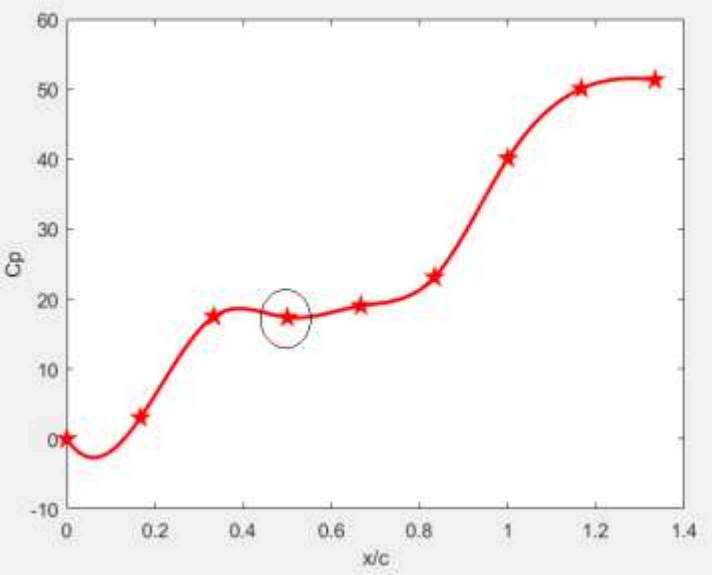

Fig-4. Cp versus x/c

7.3 Effect of variation of x/c on total pressure loss coefficient:

Figure 5. illustrates that the coefficient of total pressure loss (Cl) increases almost linearly with the increase in x/c. As distance along centreline length increases, the pressure increases and the velocity decreases. The increase in pressure causes a reduction in Cl while decrease in velocity causes an increase in Cl. However, near the inflection plane the effect is opposite as the pressure decreases and the velocity increases. These effects counteract with each other and thus produces the net effect.

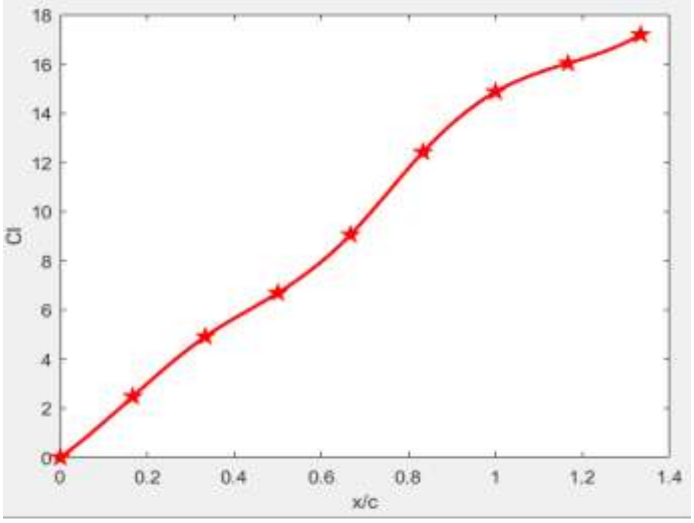

Fig-5. Cp versus x/c

7.4 Effect of wall shear stresses with X- position:

The wall shear stresses are plotted in Figure (6-8) against X-position for both the bottom and the top walls for all the diffusers to show flow separation. Separation of flow occurs due to adverse pressure gradient which is indicated by the negative or zero values of X wall shear stress.

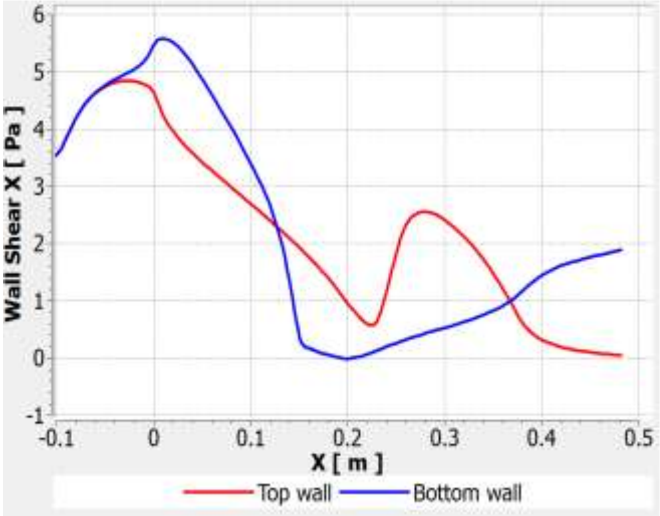

Fig- 6. Wall shear stress along X directions versus X position

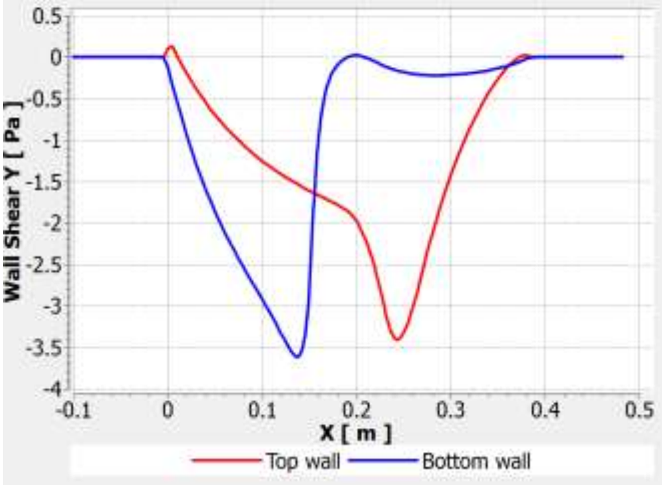

Fig-7. Wall shear stress along Y direction versus X position

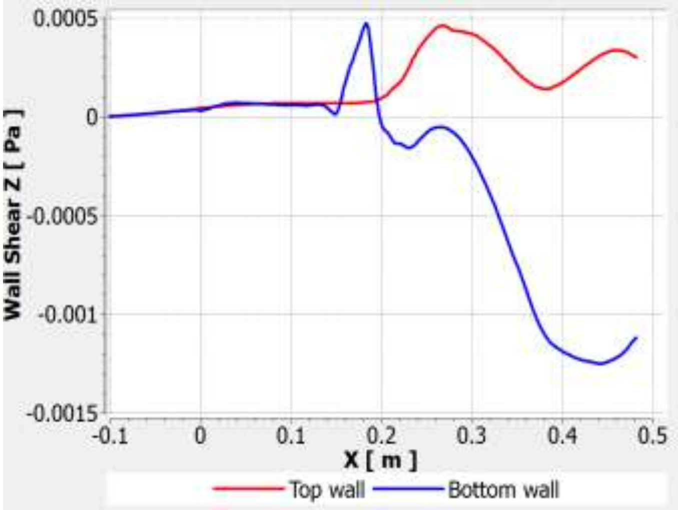

Fig-8. Wall shear stress along Z direction versus X position

8. CONCLUSIONS

Form the velocity variation and the plot of wall shear stresses it is found that the flow separation in the diffuser takes place near the bottom and top wall due to adverse pressure gradient.

Cross flow velocity vector shows the evolution of a pair of vortices rotating in the opposite direction. Due to centrifugal forces.

The overall Cp value is 51.32% of the dynamic pressure at the inlet and the overall Cl value is 17.2% of the dynamic pressure at the entry of the diffuser.

APPENDIX

Cp = total pressure loss coefficient,$(P_{ti} - P_t) / \frac{1}{2}\rho u^2_{avi}$

Cl = static pressure loss coefficient, $(P_{st} - P_{sti}) / \frac{1}{2}\rho u^2_{avi}$

c = centerline length of the duct
k = turbulent kinetic energy
x = distance along centerline
ω = specific dissipation rate

Subscripts:

t = stagnation property
st = static property
ti = stagnation property at inlet
sti = static property at inlet
i, j = tensor
avi = mass average property at inlet

REFERENCES

[1] Gupta, V., Devpura, R., Singh, S., Seshadri, V., 2001. Effect of aspect ratio and curvature on characteristics of s-shaped diffusers.

[2] Halim, M.A., Mohd, N.N., Nasir, M.M., Dahalan, M., 2018. The evaluation of k-" and k-! turbulence models in modelling flows and performance of s-shaped diffuser. International Journal of Automotive and Mechanical Engineering 15.

[3] Lee, J., Lee, S., Cho, J., 2019. Effect of inlet boundary layer suction on flow distortion in subsonic diffusing s-duct. International Journal of Aeronautical and Space Sciences, 1–8.

[4] Majumdar, B., Singh, S., Agrawal, D., 1997. Flow characteristics in the s-shaped diffusing duct. International Journal of Turbo and Jet Engines 14, 45–57.

[5] Nohseth, N.H., Nordin, N., Othman, S., Raghavan, V.R., 2014. Investigation of flow uniformity and pressure recovery in a turning diffuser by means of baffles, in: Applied Mechanics and Materials, Trans Tech Publ. pp. 526–530.

[6] Paul, A.R., Ranjan, P., Patel, V.K., Jain, A., 2013. Comparative studies on flow control in rectangular s-duct diffusers using submerged vortex generators. Aerospace Science and Technology 28, 332–343.

[7] Saha, K., Singh, S.N., Seshadri, V., Mukhopadhyay, S., 2007. Computational analysis on flow through transition s-diffusers: Effect of inlet shape. Journal of aircraft 44, 187–193.

[8] Sullerey, R., Chandra, B., Muralidhar, V., 1983. Performance comparison of straight and curved diffusers. Defence Science Journal 33, 195–203.

[9] Thenambika V, Ponsankar S, P.M.K., 2016. Design and flow analysis of s duct diffuser with submerged vortex generators. Jounal of Engineering Research and Application 6, 79–84.

BIOGRAPHIES

Manideep Roy is pursuing B.Tech. in Department of Mechanical Engineering at NIT Durgapur, India. His interest includes CFD, Fluid mechanics and Heat Transfer. He has done research internship at IIT Madras.