IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 12, Issue 3 Ver. I (May. - Jun. 2015), PP 23-28
www.iosrjournals.org

Performance Improvement of S-Shaped Diffuser Using Suction and Blowing Methods

[1]Manoj Kumar Gopaliya, [2]Amit Singla, [3]Amulya Saini.
[1]Assistant Professor, Department of Mechanical Engineering, ITM University, Gurgaon Sector-23 A, Gurgaon-122017
[2&3] B. Tech. Scholar, Department of Mechanical Engineering, ITM University, Gurgaon Sector-23 A, Gurgaon-122017

Abstract: *This paper presents analysis of suction and blowing techniques for performance improvement of S-shaped diffuser of 90°/90° turn with rectangular inlet and outlet using Computational Fluid Dynamics (CFD). A finite volume based technique using k–ε turbulence model has been adopted and modified to predict the flow and to analyze the variation in different flow parameters due to these techniques. The present work shows around 8% improvement in pressure recovery of the diffuser due to Suction Method whereas a marginal improvement of around 3% is achieved through Blowing Method.*

Keywords: *Blowing, boundary layer, CFD, diffuser, non-uniformity, outlet pressure recovery, S-shaped diffuser, suction, turbulence.*

Nomenclature

AR	Area ratio	Re	Reynolds number $Re = \rho U_{avgi} D_h/\mu$
AS	Aspect ratio	D_h	Hydraulic Diameter at the inlet where D_h = 4xArea/Perimeter
X, Y, Z	Cartesian coordinates	P_{sx}	Mass average static pressure at any location "x"
k	Turbulent kinetic energy	P_{se}	Mass average static pressure at exit
B	Width at outlet	P_{si}	Mass average static pressure at inlet
ε	Rate of turbulent kinetic energy dissipation	P_{tx}	Mass average total pressure at any location "x"
ρ	Fluid density	P_{te}	Mass average total pressure at exit
μ	Viscosity of working fluid	P_{ti}	Mass average total pressure at inlet
ξ_o	Effectiveness of diffuser	S_{ix}	Non-uniformity index at any location "x"
C_{psx}	Coefficient of static pressure gain at any location "x"	U_{avgx}	Mass average mean velocity at any location "x"
C_{pse}	Coefficient Static pressure gain at exit	U_{avgi}	Mass average mean velocity at inlet
$C_{pse(ideal)}$	Coefficient of static pressure gain ideally $C_{pse(ideal)} = 1 - 1/AR^2$	V_x	Velocity in X-direction
C_{ptx}	Coefficient of total pressure loss at any location "x"	V_y	Velocity in Y-direction
C_{pte}	Coefficient of total pressure loss at exit	V_z	Velocity in Z-direction

I. Introduction

A diffuser is a mechanical device which recovers the pressure energy from the flowing fluid at the expense of its kinetic energy. Diffusers are of many types namely axial, radial and curved depending on the geometry and design and find very wide applications in the field of turbo-machinery and aerospace. Curved diffusers are finding huge applications in the field of aircraft applications due to space restrictions and design compatibility. Study of flow parameters within these diffusers has been of primary interest to researchers in the area of fluid mechanics right from the start of 21st century. Pioneer in this was Eustine (1910) who has investigated the nature of flow within the curved diffusers. The continuous, committed and collective efforts of researchers thereafter have helped in the understanding of the fluid mechanics within these diffusers. With ever increasing the computational capability and development of very strong Computational Fluid Dynamics (CFD) codes have results in increased research activities in this area in recent times.

S-shaped diffusers are one of the popular types of curved diffusers. Generation of very strong pressure driven stream wise vortices due to inflexion in the curvature along the direction of flow make flow in these diffusers very complicated. Since now, many researches have been carried out on S-shaped diffuser dealing with issues like effect of different inlet conditions, effect of angle of attack, effect of area ratio, effect of aspect ratio, effect of different inlet and outlet cross-section geometries, effect of angle of turn and many more on pressure recovery and non-uniformity of flow through it. Notable among them, Majumdar (1994) and Majumdar et al. (1997; 1998) have made detailed measurements on S-shaped diffuser of rectangular cross-section having an area ratio 2 and inlet aspect ratio 6. Anand et al. (2001; 2001) have performed experimental investigations on circular (22.5^o/22.5^o) and rectangular (90^o/90^o) diffusers having area ratio (AR) 1.9 and 2 respectively.

Investigators are also showing keen interest towards automated design and optimization of S-ducts. Zhang et al. (2000) has developed an optimization theory for 3-D S-diffuser in subsonic range to reduce total pressure distortion and sustain total pressure recovery. Lefantzi and Knight (2002) have developed an automated design optimization process for S-diffuser taking into account constraints like airframe weight, space and line-of-sight blockage of the engine face.

Apart this, many researchers have significantly contributed in the area of performance-improvement of S-duct by secondary flow control. A remarkable improvement in the performance of rectangular cross-sectional diffuser (AR = 4) with higher diffusion angle using suction technique to control the boundary layer separation has been achieved by Yoshimasa et al. (1970). Chen and He (1991) have showed reduction in separation region resulting improved diffuser performance of a highly curved S-shaped subsonic diffuser with high aspect ratio by selecting proper geometric parameters and location of the vortex generator. Other than this, Reichert and Wendt (1994) have also used vortex generators but of fin-type for secondary flow control and has found considerable improvement in various flow parameters of the diffuser. Other notable research in this area is of Pradeep and Sullerey (2004) who has adopted vortex generator jets for the above purpose and has found 20% decrease in total pressure loss and flow distortion coefficient in the diffuser. Another performance improvement technique adopted by some of the researcher for curved diffusers is momentum injection. Singh et al. (2009) and Gopaliya et al. (2011) have done some parametric study using this technique on S-shaped diffusers in which a rotating cylinder is placed at the inflexion plane for secondary flow control by imparting momentum to the retarding working fluid.

The present work also contributes in this area of research by providing analysis of suction and blowing methods for performance improvement of curved diffusers.

II. Geometrical Description And CFD Analysis Of S-Shaped Diffuser

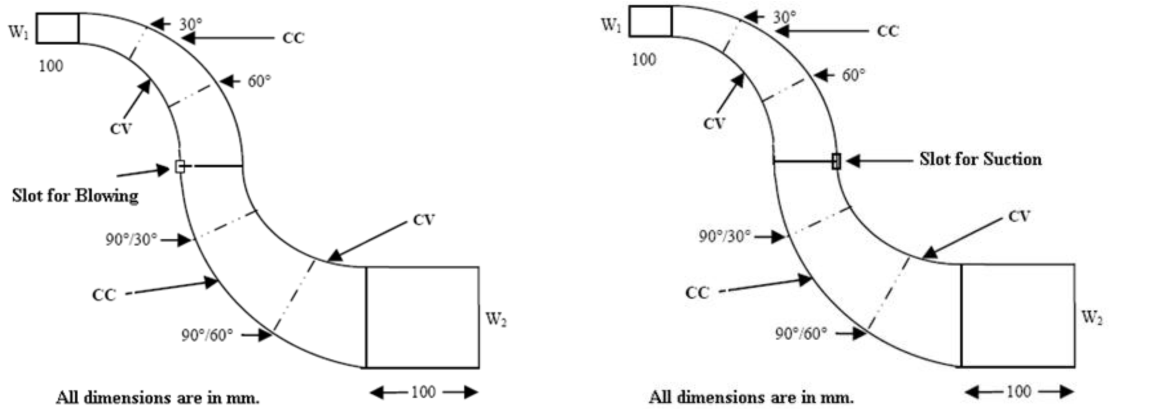
2.1 Geometrical Parameters

S-shaped diffuser used during this research work is same as what used by Majumdar (1994). It has a 90°/90° turn. The radius of curvature for both upper and lower limbs is 191 mm at the centerline and the area ratio of 2 is maintained at the diffuser outlet. Straight portion of 100 mm long is provided at both ends of the diffuser for enabling flow settlement. A provision of blowing and suction at the inner and outer inflexion planes of this duct has been made in the form of thin slot across whole breadth.

2.2 Description of Geometric Model Generation and Computational Tools Used

The geometric model has been created using GAMBIT (Active Bangalore; 2009) modelling software using dimensions shown in **Figure 1**. S-duct has been made by using surfacing technique with the help of sweeping solid command. Slots with different opening sizes ranging from 5 to 20 mm have been provided on the inner section of the duct for blowing purpose. Similarly, slots on the outer section of the duct have been provided for suction. The flow domain is divided into to parts for meshing purpose; one near the boundary where very fine boundary layer meshing has been made to cater for steep velocity gradients and rest portion has been meshed using hexahedral map meshing scheme.

Grid independency check has been performed by carrying out simulations over four different mesh sizes with total number of control volumes equal to 307530, 402230, 537290 and 776020 respectively. Since no significant change in results has been noticed in the last two meshes, mesh with 537290 control volumes has been finalized for further analysis.



W1 = 50 mm, W2 = 100 mm Length (L) = 600 mm, breadth (b) = 300 mm
AR = W2/W1 = 2, AS = b/W1 = 6.0, L /W1 = 12.0

W1 = 50 mm, W2 = 100 mm Length (L) = 600 mm, breadth (b) = 300 mm
AR = W2/W1 = 2, AS = b/W1 = 6.0, L /W1 = 12.0

Figure 1: Geometry of S-Shaped Diffuser

A commercial CFD code FLUENT 6.3 (Active Bangalore; 2009) has been used for the analysis of flow through this diffuser. The governing equations (Launder and Spalding; 1974) for averaged flow in reduced form for steady incompressible flows are:

$$\frac{\partial(\rho u_i)}{\partial x_i} = S_m \qquad (1)$$

$$\rho u_j \frac{\partial u_i}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) - \frac{2}{3}\,\mu\delta_{ij}\frac{\partial u_i}{\partial x_i}\right] + \frac{\partial}{\partial x_j}\left(\overline{-\rho u_i' u_j'}\right) \qquad (2)$$

These equations are of the same general form as the original equations except for some additional terms, called Reynolds stresses or turbulent stresses, which are modelled using closure equations. In present research, turbulence model Standard k-ε with boundary conditions prescribed in **Table 1** has been adopted during simulations.

Table 1: Description of boundary conditions

Inlet boundary conditions:		
i	Type of boundary	**Velocity-inlet**
ii	Reynolds number	**2.35×10^5**
iii	Turbulence intensity	**10%**
Outlet boundary conditions:		
i	Type of boundary	**Pressure-outlet**
ii	Pressure-specified	**0 Pa (Gauge)**
Wall boundary conditions:		
i	Type of boundary	**Rough; 0.02 mm Roughness height**
ii	Shear condition	**No-slip**
Working fluid properties:		
i	Working fluid	**Air**
ii	Density of working fluid	**1.225 kg/m^3**
iii	Viscosity of working fluid	**1.7894×10^{-5} kg/m-s**
Element shape:		**Tetrahedral**
Number of cells:		**537290 volume cells**
Constants: $C_{1\varepsilon}$ = 1.44, $C_{2\varepsilon}$ = 1.92, C_μ = 0.09, σ_k = 1.0 and σ_ε = 1.33		

To calculate the Reynolds stresses using Standard k-ε model, an extended Boussinesq relationship has been adopted as follows:

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

where μ_t is the eddy viscosity, k is the turbulent kinetic energy and δ_{ij} is the Kronecker delta.
μ_t is calculated as follows:

$$\mu_t = \frac{\rho C_\mu k^2}{\varepsilon} \qquad (4)$$

For steady incompressible flow, additional equations used for k and ε are presented below:

$$\rho u_i \frac{\partial k}{\partial x_i} = \frac{\partial}{\partial x_i}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_i}\right] + G_k - \rho\varepsilon + Y_M \qquad (5)$$

$$\rho u_i \frac{\partial \varepsilon}{\partial x_i} = \frac{\partial}{\partial x_i}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_i}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}G_k - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \qquad (6)$$

G_k represents the production of turbulent kinetic energy and is calculated as:

$$G_k = \mu_t S^2 \qquad (7)$$

where S is the modules of mean rate of strain tensor defined as:

$$S = \sqrt{2S_{ij}S_{ij}} \qquad (8)$$

Here mean strain rate S_{ij} given by:

$$S_{ij} = \frac{1}{2}\left[\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right] \tag{9}$$

The dilatation dissipation term (Y_M) has been neglected for incompressible flow. The value of the empirical constants used are $C_{1\varepsilon} = 1.44$, $C_{2\varepsilon} = 1.92$, $C_\mu = 0.09$, $\sigma_k = 1.0$ and $\sigma_\varepsilon = 1.33$. These values have been found to work reasonably well for a wide range of wall bounded and free shear flows.

Flow simulations in FLUENT has been carried out adopting a cell centred finite volume method to solve various partial differential equations using segregated solver. An implicit solution scheme along with an algebraic multi-grid (AMG) method has been adopted for faster convergence. The second-order upwind scheme has been used for discretization of all equations to achieve higher accuracy. Coupling between velocity and pressure has been done by pressure velocity correlation using SIMPLE algorithm. Under relaxation factors were used for all equations to satisfy the Scarborough condition for convergence of the solutions and also avoiding the divergence in the solutions. Convergence of the solution was assumed when sum of normalized residual for each conservation equation was less than or equal to 10^{-6}. The residuals for different equations are given as:

$$\frac{\sum_{i=1}^{M} |R_i|^n}{S_{N\phi}} \leq 10^{-6} \tag{10}$$

where $|R_i|^n$ is the sum of the absolute residuals for a dependent variable ϕ for the n^{th} iteration and $S_{N\phi}$ the corresponding normalizing factor.

2.3 Performance Parameters

The important performance evaluation parameters are:

a.) Coefficient of static pressure gain: It represents the extent by which kinetic energy has been converted into pressure energy due to diffusing action at any location (say x).

$$C_{psx} = 2(P_{sx} - P_{si})/\rho U^2_{avgi}$$

b.) Effectiveness of diffuser: It represents the pressure recovery capacity of diffuser in comparison with ideal diffuser.

$$\xi_o = C_{pse}/C_{pse(ideal)}$$

c.) Coefficient of total pressure loss:

$$C_{pt} = 2(P_{ti} - P_{tx})/\rho U^2_{avgi}$$

d.) Non-uniformity index: It represents the percentage amount of secondary flow at any location.

$$S_{ix} = 100*(\sum \text{sqrt}\,(V_y^2 + V_z^2))/\,(n*U_{avgx})$$

Here "summation sign" includes all the grid point in the outlet plane and "n" is its numerical value.

2.4 Convergence Criteria and Validation of CFD Code

The model and boundary conditions (**Table 1**) have been validated with the experimental values obtained by Majumdar (1994) which are in good agreement (**Figure 2**). A convergence criterion of 10^{-6} has been adopted during the simulations.

It was shown by Gopaliya et al. (2011) in the study of combined offset effect using another S-shaped diffuser that among two very popular turbulent models i. e. Standard k–ε turbulence model and Standard k–ω turbulence model, later gives higher deviations from experimental data during flow analysis through the duct. This is due to the fact that the k–ω turbulence model produces best results for low Reynolds number flows; however the Reynolds numbers involve in present study are very high. Thus standard k–ε turbulence model has been chosen in the present.

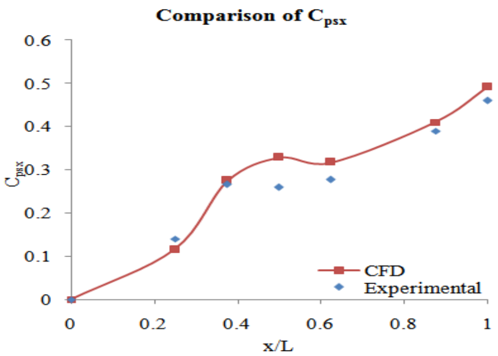


Figure 2: Comparison of C_{psx} over the length of the diffuser

The details regarding wall pressure distribution, location of separation & concentration of secondary flow throughout the diffuser have not been presented in here. However, during validation process these aspects were kept into consideration and therefore the meshing intensity close to the walls were increased to resolve the lower sub-layer (y+) values.

The deviations have not been resolved completely, which may be attributed to the fact that the flow is highly 3D and there is always some uncertainty exists in the experimental values due to limitations of various instruments used for measurements. The second reason could be that the k-ε turbulence model has its limitation in resolving the flow in the region having sharp variations in velocity gradient.

III. Results And Discussions

3.1 Suction Method

In this technique the slots of different depths ranging from 5 to 20 mm for providing suction have been made on the outer section of the duct as shown in **Figure 1**. The optimised depth of suction slot is obtained by analysing change in outlet pressure recovery with change in depth of the suction slot at any particular suction pressure which in present case is -350 Pa (**Table 2**). The slot of 20 mm depth is finalized for further simulations. Simulations on the duct with 20 mm suction slot have been carried out for suction pressure ranging from -400 Pa to -1200 Pa. The variations in Coefficient Static pressure gain at exit, non-uniformity index at exit and loss in mass flow rate with suction pressure have been plotted in **Figure 3**.

Table 2: Variation of outlet pressure recovery with depth of suction slot

S. No.	Depth of suction slot (in mm)	C_{pse} (in %)
1	Without slot	46*
2	5	48.54
3	10	50.63
4	15	52.04
5	20	53.71
6	25	52.91

*Experimental value obtained by Majumdar [2].

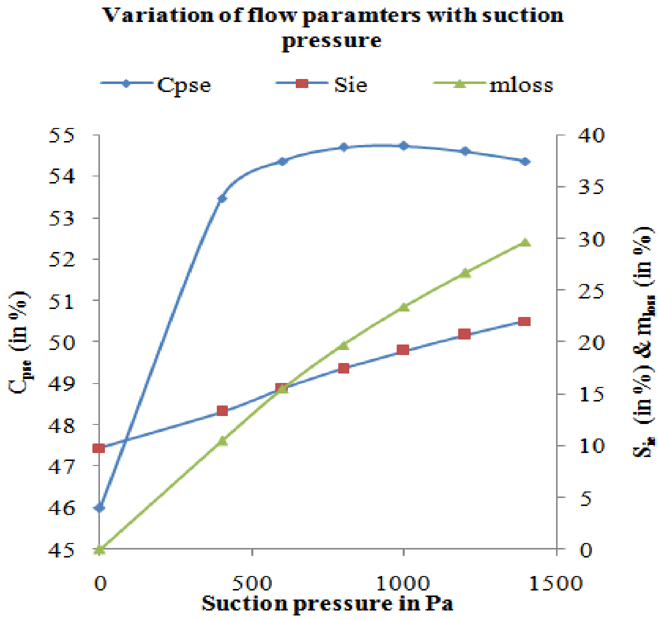


Figure 3: Variation of flow parameters with suction pressure

It has been observed that the pressure recovery improves with increase in suction pressure upto -1000 Pa and then drops. However, non-uniformity in flow and loss in the mass flow rate at the diffuser exit increases beyond -1000 Pa as well. Hence, suction pressure of -1000 Pa is found to be the most suitable for diffuser in consideration. Suction method results in improvement in performance of the diffuser due to removal of detached fluid layer from the duct at the inflexion point. However, beyond certain value of suction pressure, further increase will results in removal of energised fluid layers as well leading to drop in the performance; which is quite evident in the present research as well where the pressure recovery drops after -1000 Pa of suction pressure.

3.2 Blowing Method: In this technique, a slot of 5 mm has been created at the inner section of the duct as shown in **Figure 1**. Angle at which the velocity is blown in through the slot is optimised by analysing the pressure recovery at the diffuser exit for a fixed blowing velocity of 30 m/s with different angle of attack. Angle of attack of 80^o has been finalized as the pressure recovery is found to be the maximum at this angle (**Table 3**). The coefficient static pressure gain at exit is found to be 49.42 for 30 m/s of blowing speed at 80^o of angle of attack which is marginally better that the pressure recovery without any method.

Table 3: Variation of outlet pressure recovery with angle of attack

S. No.	Angle of attack	C_{pse} (in %)
1	Without blowing	46*
2	50	47.6
3	55	48.2
4	58.7	48.53
5	65	48.98
6	70	49.21
7	75	49.35
8	80	49.42

*Experimental value obtained by Majumdar [2].

It is quite evident that the blowing method has failed to improve the performance of the given diffuser significantly. It may be attributed to increased disturbances at the infexion plane due to blowing which in place of energzing the retarded layer is actually leading to more seconary losses.

IV. Conclusions

This paper presents analysis of suction and blowing techniques for performance improvement of S-shaped diffuser of 90°/90° turn with rectangular inlet and outlet using CFD.
The following specific conclusions have been drawn from present study:

i. Suction slot of 20 mm is best suited for creating suction in the given diffuser.
ii. Suction pressure of -1000 Pa gives maximum pressure recovery in the diffuser under consideration.
iii. Pressure recovery of the diffuser improves by around 8% due to suction method.
iv. An angle of attack of 80^o is best suited for blowing in the given diffuser.
v. Pressure recovery of the diffuser improves marginally by around 3% due to blowing method

References

[1]. Eustine, J. (1910) ‘Flow of Water in Curved Pipes’, Proc. Royal Society, London, Series A – 84, pp.107-118.
[2]. Majumdar, B. (1994) ‘Flow Investigation in Curved Diffuser’, Ph.D. Thesis, Department of Applied Mechanics, Indian Institute of Technology Delhi, New Delhi, India.
[3]. Majumdar, B., Singh, S. N. and Agrawal, D. P. (1997) ‘Flow Characteristics in S-Shaped Diffusing Duct’, International Journal of Turbo and Heat Engines, Vol. (14), pp. 45-47.
[4]. Majumdar, B., Ratan Mohan, Singh, S. N. and Agrawal, D. P. (1998) ‘Experimental Study of Flow in High Aspect Ratio 90° Curved Diffuser’, Transaction ASME, Journal of Fluid Engineering, Vol. (120), pp. 83-89.
[5]. Anand, R. B., Lajpat Rai, Singh, S. N. and Sharma O. P. (2001) ‘Flow Characteristics of A Low Aspect Ratio 90°/90° S-Shaped Diffuser’, Journal of Aeronautical Society of India, Vol. (53), pp. 239-252.
[6]. Anand, R. B., Lajpat Rai and Singh, S. N. (2001) ‘Flow and Performance Characteristics of 22.5°/22.5° S-Shaped Circular Diffuser’, Proceedings of 28th national conference on FMEP, Chandigarh, India, pp. 364-371.
[7]. Zhang, W., Knight, D. and Smith, D. (2000) ‘Automated Design of A Three Dimensional Subsonic Diffuser’, Journal of Propulsion and Power, Vol. (16), No. 6, pp. 1132-1140.
[8]. Lefantzi, S. and Knight, D. (2002) ‘Automated Design Optimization of A Three Dimensional S-Shaped Subsonic Diffuser’, Journal of Propulsion and Power, Vol. (18), No. 4, pp. 913-921.
[9]. Yoshimasa F., Tesuo F., Eisho Y., Tsuzuki I. and Ichiro N. (1970) ‘Performance of the two dimensional diffuser with suction at the entrance’. Bulletin of JSME, 13, pp. 264–271.
[10]. Chen X. and He W. (1991) ‘Flow characteristics and flow control for high aspect ratio and highly curved S-bend diffuser’, Journal of Propulsion Technology, 60, pp. 17–22.
[11]. Reichert, B. A. and Wendt, B. J. (1994) ‘Improving Diffusing S-Duct Performance by Secondary Flow Control’, 32nd Aerospace Sciences Meeting and Exhibit, NASA Lewis Research Center, Cleveland, Ohio, AIAA Journal.
[12]. Pradeep, A. M. and Sullerey, R. K. (2004) ‘Secondary Flow Control in A Circular S-Duct Diffuser Using Vortex Generator Jets’, 2nd AIAA Flow Control Conference, 28 June–1 July, Portland, Oregon.
[13]. Singh, S. N., Seshadri, V., Chandel S. and Gaikwad M. (2009) ‘Analysis of the Improvement in Performance Characteristics of S-Shaped Rectangular Diffuser by Momentum Imparting Using Computational Fluid Dynamics’, Engineering Applications of Computational Fluid Mechanics, Vol. 3, No. 1, pp. 122-135.
[14]. Gopaliya, M. K., Jain, P., Kumar S., Yadav V. and Singh S. (2014) ‘Performance Improvement of S-Shaped Diffuser Using Momentum Imparting Technique’, Volume 11, Issue 3, Version I, pp. 23-31.
[15]. Active Bangalore (2009). FLUENT 6.3 User Guide, Vol. 1–4.
[16]. Launder, B. E. and Spalding, D. B. (1974) ‘The numerical computation of turbulent flow’, Computational Methods in Applied Mechanics and Engineering, Vol. (3), Issue 2, pp. 269-289.
[17]. Gopaliya, M. K., Goel, P., Prashar, S. and Dutt, A. (2011) ‘CFD analysis of performance characteristics of S-shaped diffusers with combined horizontal and vertical offsets’, Computers & Fluids, Elsevier Publications, Vol. (40), pp. 280–290.