CFD Modeling of Flow through S-Shaped Duct with Tangential Blowing

Md. Nadim Shams, Raj Kumar Singh, M. Zunaid*, Md. Gulam Mustafa

Delhi Technological University, Bawana Road, Delhi, India
E-mail: mzunaid3k@gmail.com*

Abstract: This Paper investigates the flow inside an S-Shaped Square Duct. In this paper, a computational fluid dynamics model of fully developed turbulent flow (k-ε model) is implemented with the help of FLUENT software and the variation of pressure along the length of bend with variation in Reynolds number is analyzed. The curvature is investigated at Reynolds numbers $4.73\text{x}10^4$ and $1.47\text{x}10^5$. A non-dimensional parameter ω, defined as the total pressure loss coefficient is analyzed for finding out the total pressure loss in the duct. Cp Data obtained from the simulation of S-shaped ducts show that there is flow separation at the near side wall of the first bend and far side wall of the second bend. At high Reynolds number separation is more dominant near junction of bend as compare to low Reynolds number flow. To improve the flow in S-ducts, Tangential Blowing as flow control methods is implemented. The method used is effective in suppressing flow separation and reducing total pressure loss.

Keywords: S-shaped Duct, Tangential Flow, CFD, Fluent, Ansys, Reynold number

I. INTRODUCTION

The layout of any practical piping system necessarily includes bends and the accurate prediction of pressure losses, flow rate and pumping requirements demands knowledge of the character of curved duct flows, such wide applications in the industry have forced researchers to acknowledge the importance of study of flow in curved ducts.

Curved duct flows are common in aerospace applications. Many military aircraft have wing root or ventral air intakes and the engine is usually positioned in the Centre of the aircraft's fuselage. Air entering these intake ducts must be turned through two curves (of opposite sign) before reaching the compressor face. Such a configuration results in an S-shaped air intake duct and therefore the engine performance becomes a strong function of the uniformity and direction of the inlet flow and these parameters are primarily determined by duct curvature.

Shams et al. [18] study is based on flow inside S-shaped square duct which model in gambit software using k-ε model. From simulation, it is found out that flow separation is at near side of first bend and far side of second bend. Vortex generator method is used to improve the flow

This chapter intends to present a review of the flows in curved ducts and S-shaped ducts. Discussion is based on the Flow Separation, mechanism of total pressure loss and duct's exit flow conditions.

II. RESULTS AND DISCUSSION

The optimum aerodynamic performance of S-shaped ducts (or aircraft air-intake ducts) demands that a relatively uniform flow with a smallest possible pressure loss. These requirements naturally lead one to consider the use of traditional flow control devices like vortex generators, blowing jets. Passive devices like Tangential Blowing takes place on the side wall eliminate flow separation if it is present. They “locally” mix the high-momentum fluid in the free stream with low-momentum fluid near the wall and thus suppress flow separation. In contrast to the passive means of flow control, blowing jets and vortex generator jets are active flow control devices for flow separation control, whereby mass addition near the separation point energizes the low momentum fluid close to the wall to overcome the adverse pressure gradient.

To study the effects of flow control in square cross sectioned S-duct, Tangential Blowing flow control method was studied In this Chapter, the relative merit of vortex generators on the flow in a square cross sectioned S-duct. Suppression of flow separation, reduction of total pressure loss, and flow uniformity at duct exit are the chosen criteria.

Investigation of (S-duct Figure 1 [18]) has been carried out with help of FLUENT, a CFD tool to simulate the Effect of pressure is studied. The present work is conducted at higher *Re*=$4.73 \times 10^4$and $1.47 \times 10^5$and with square cross-sectioned, S- shaped ducts with sharper bends and larger turning angle. The geometry is made in solid works and imported on Ansys software for analysis. After analysis on FLUENT following graphs are plotted.

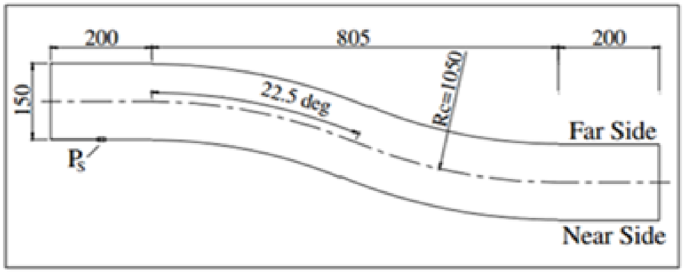

Figure 1. S-duct configuration

Case 1: Study of 3D S-Shaped Duct:

Mesh Independency Study:

The grid independency is studied for the k-ε model employing four size of grid to examine the sensitivity of grid. As we get result in the third column that is fine mesh size is independent of grid that’s the optimum grid.

On the study of 3d s-shaped duct we found that from Fig.2 and Fig.3 i.e. contour of static pressure there are two low pressure zones are creating on the duct one at near side wall of the 1st bend and other on the far side wall of the second bend. On these two pocket of the bend there is a flow separation which is clearly seen by the Fig. no.4 i.e. contour of velocity vector. For suppressing this flow separation, we can use different flow control techniques. From Fig. 2 and Fig. 3 it is clearly seen that the contour of static pressure is same for both the Reynolds number only there is difference in magnitude. Fig. 5 shows that the plot of static pressure vs. position.

Table 1. Grid independency chart

Mesh size	Maximum cell squish	Maximum aspect ratio	Pressure and static pressure(min and max)	Pressure and pressure coefficient
coarse	5.05561×10^{-2}	2.03094	-1.37609, 3.683053	-2.246697, 6.013155
medium	6.5253×10^{-2}	2.16342	-1.553137, 3.96047	-2.535753, 6.466074
fine	9.28082×10^{-2}	2.18632	-1.585209, 4.26690	-2.5881, 6.966368
Mesh with sizing	9.28082×10^{-2}	2018632	-1.585209, 4.26690	-2.5881, 6.966368

Graphs of Static Pressure contours

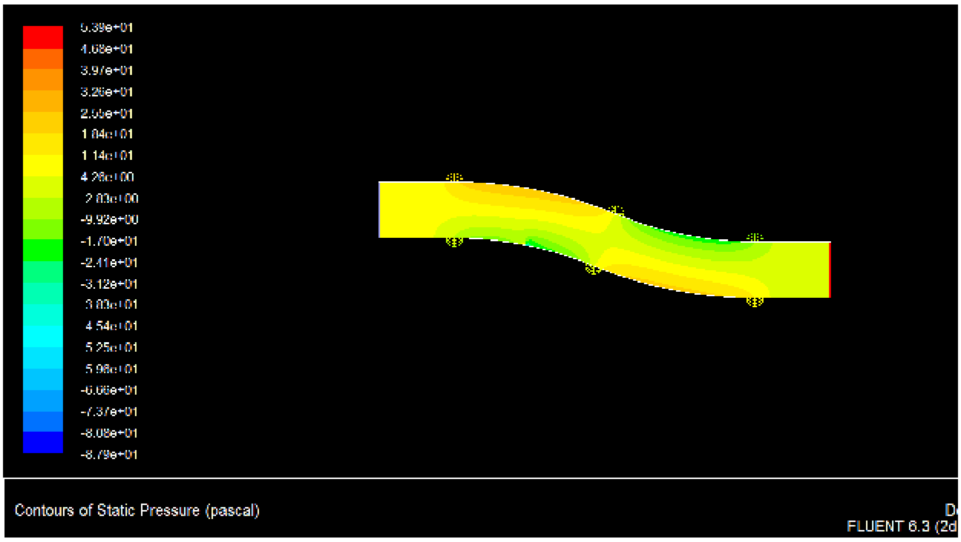

Figure 2. Contour of Static Pressure at Re=4.73×10^4

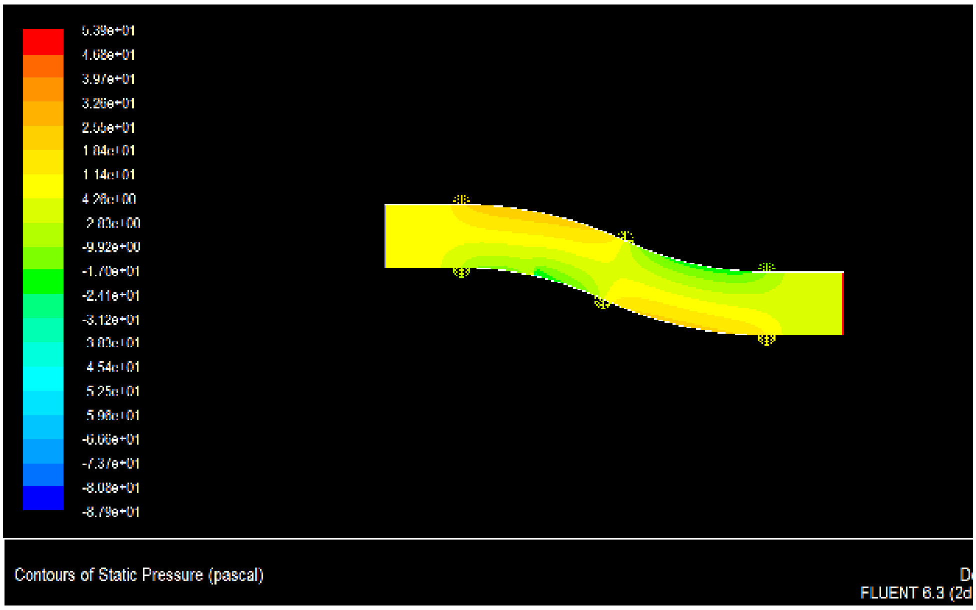

Figure 3. Contour of static Pressure at *Re*=1.47×10^5

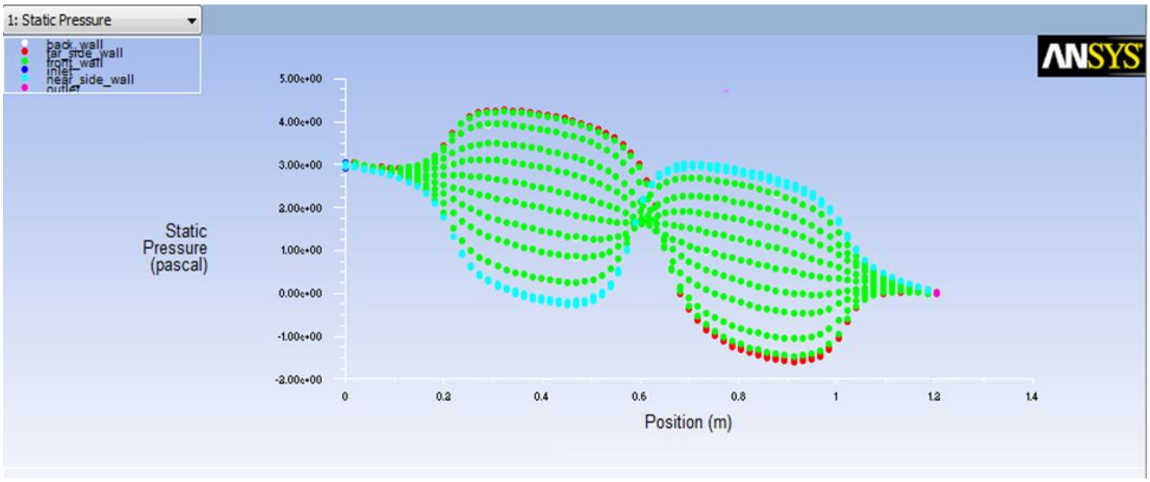

Figure 4. Plots of static pressure v/s position

Study of 3D S-Shaped Duct with Flow Control Techniques:

Different flow control Techniques for improving the performance of S-Duct.

- Tangential Blowing

1. Study of 3D S-Shaped Duct with Tangential Blowing:

Vortex generators "locally" mix the high momentum fluid in the free stream with the low momentum fluid near the wall and thus energies the boundary layer to suppress flow separation.

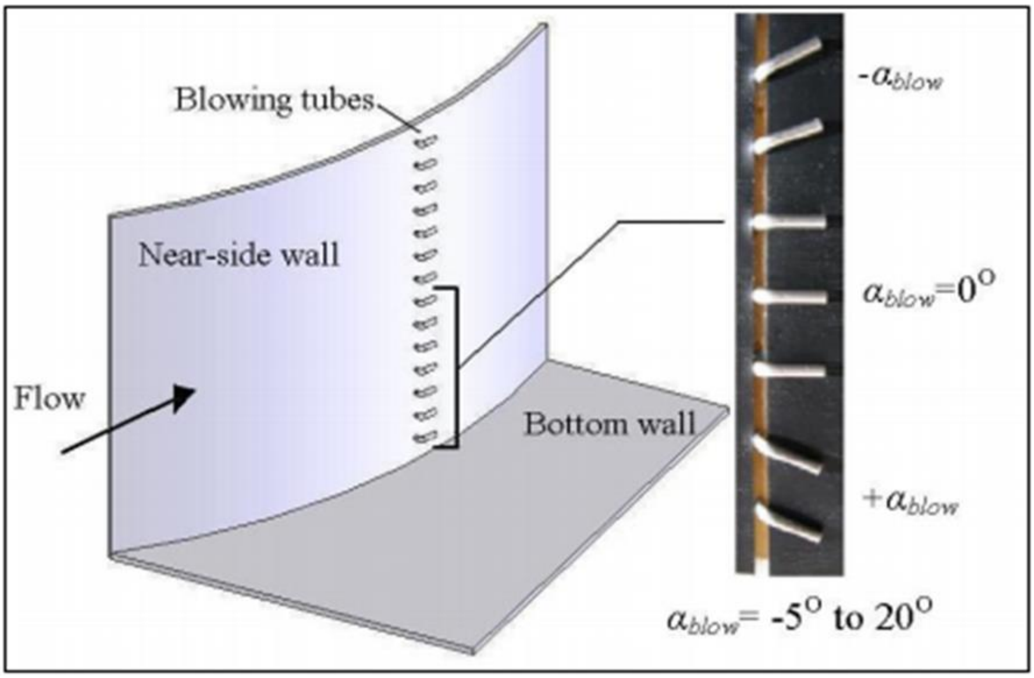

Figure 5. Tangential Blowing on near sidewall

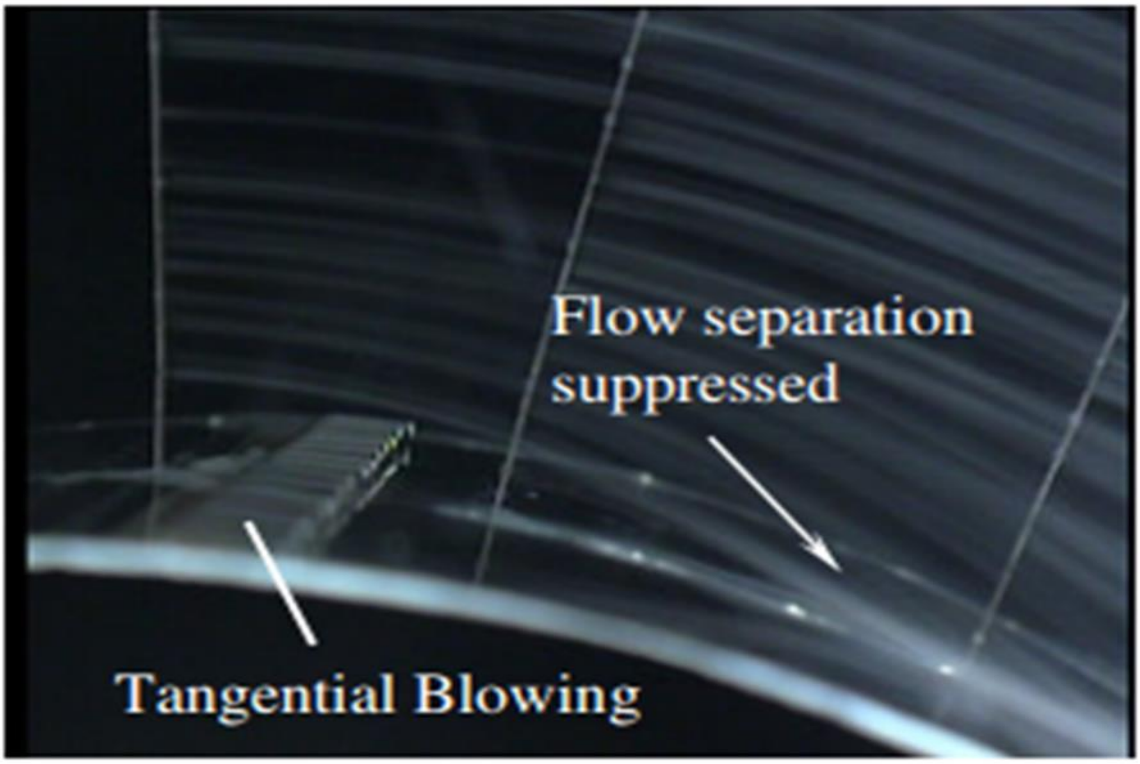

Figure 6. Shows flow separation and suppressed flow

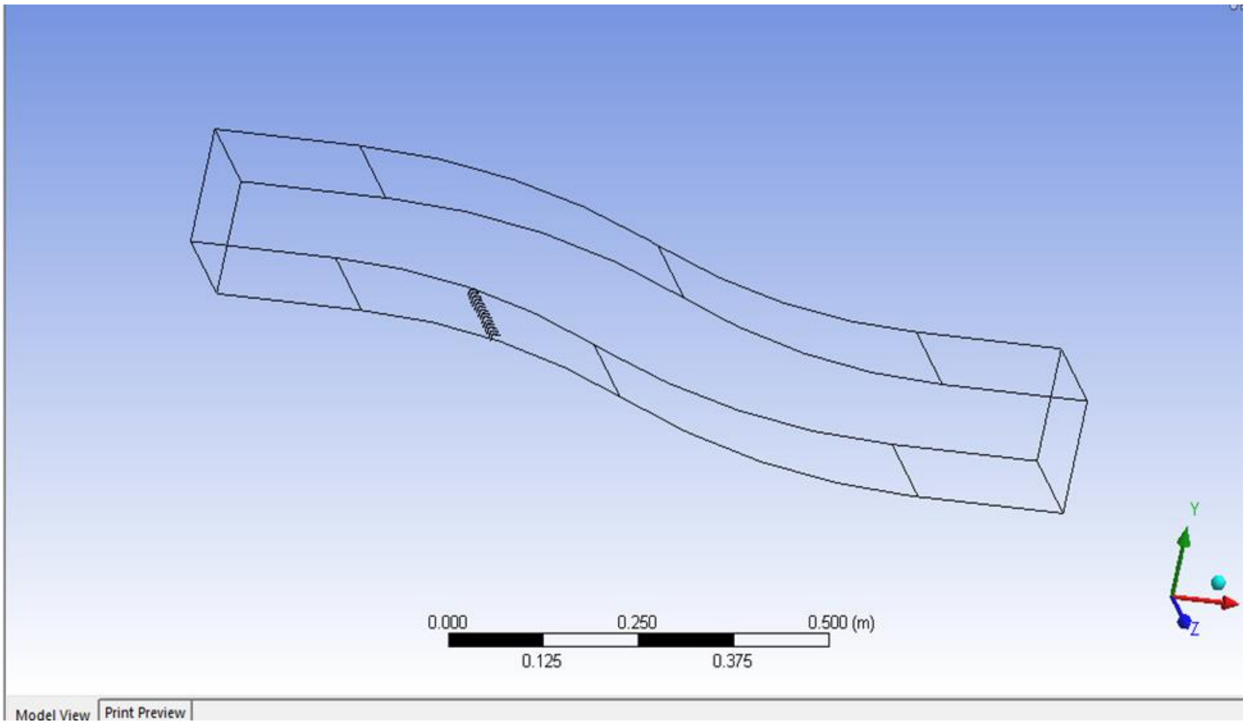

Figure 7. S-duct with Tangential Blowing

On the study of 3D S-shaped duct with flow control technique with Tangential Blowing, we found that from Figure 2 and Figure 3, contour of static pressure there are no low-pressure zones create on the duct one at near sidewall of the 1st bend and other on the far sidewall of the second bend as found from the analysis from bare duct. From Figure 2 and Figure 3, it is clearly seen that the contour of static pressure is same for both the Reynolds number only there is difference in magnitude. Figure 4 shows that the plot of static pressure vs. position the trend obtained from Figure 4 is clearly matched with the Figure 9 i.e. practically result obtained by the Thye, N. Y., 2009.now from the different values obtained from the pressure graph we calculate value of Cp at different point of the duct which is shown on the Table no 2. Fig.12 shows graph obtained Cp v/s s/d by simulation which is matched by the practical result.

From the above simulation results following tables and graphs are plotted for surface pressure distribution Cp and total pressure loss coefficient.

1. Table of surface pressure distribution on the side wall (far side wall and near side wall) for bare duct and duct with Tangential Blowing at Re=4.73x10^4

Table.2: Surface pressure distribution on the side wall at Re=4.73x10^4

S/D	Cp far Side (bare)	Cp far Side (blowing)	Cp Near Side (bare)	Cp Near Side (blowing)
0	0.2	0.25	-0.3	-0.35
0.25	0.3	0.3	-0.45	-0.47
0.5	0.35	0.37	-0.5	-0.71
0.75	0.43	0.44	-0.55	-0.9
1	0.3	0.35	-0.4	-0.55
1.25	0.3	0.3	-0.5	-0.3
1.5	-0.3	-0.1	-0.4	-0.1
1.75	-0.75	-0.25	0	0.3
2	-1	-0.6	-0.1	0.3
2.25	-1.2	-0.9	0.05	0.4
2.5	-1.2	-0.8	0.1	0.4
2.75	-1	-0.8	0.15	0.25
3	-0.9	-0.65	0.1	0.25
3.25	-0.5	-0.6	0.2	0.2

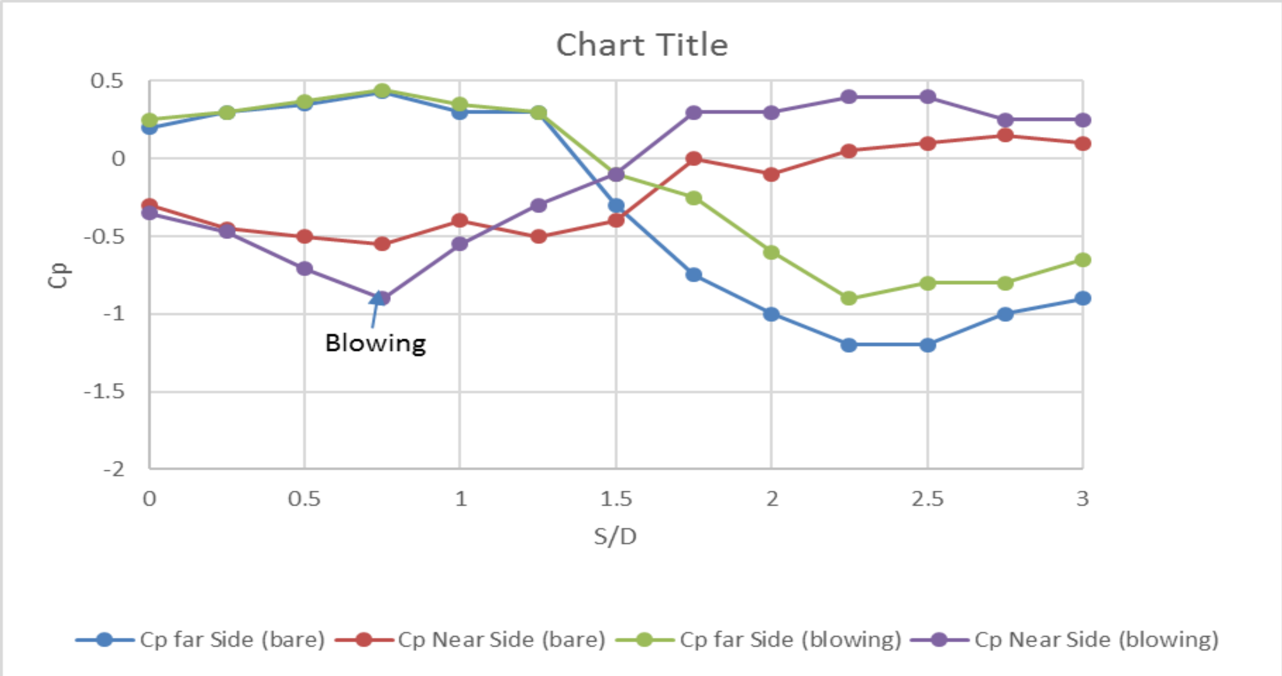

Figure 8. Surface pressure distribution on the side wall at Re=4.73x10[4]

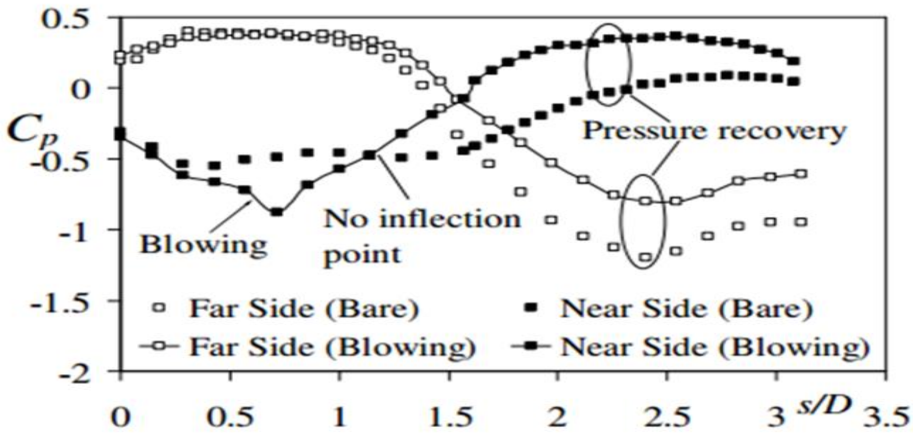

Figure 9. Graph of Cp v/s S/D obtained practically by Thye, N. Y [13] at Re=4.73x10[4]

From above chart it is clearly seen that there is no point of inflection in case of duct with Tangential Blowing as compare to bare duct so it can be concluded from above chart that flow separation is suppressed by use of flow control device i.e. Tangential Blowing.

Figure 8 and Figure 10 shows that there is no point of inflection in case of duct with Tangential Blowing as compare to bare duct. So it can be concluded from above chart that flow separation is suppressed by use of flow control device i.e. Tangential Blowing and the pattern obtained by simulation is similar to the graph of Cp vs S/D obtained practically by Thye, N.Y. i.e. Figure 9.

2. Table of surface pressure distribution on the side wall (far side wall and near side wall) for bare duct and duct with Tangential Blowing at *Re*= 1.47×10^5

Table 3: surface pressure distribution on the side wall at *Re*=4.73x10^4

S/D	Cp far Side (bare)	Cp far Side (blowing)	Cp Near Side (bare)	Cp Near Side (blowing)
0	0.25	0.25	-0.25	-0.35
0.25	0.3	0.3	-0.45	-0.47
0.5	0.45	0.37	-0.67	-0.71
0.75	0.43	0.44	-0.55	-0.9
1	0.4	0.42	-0.5	-0.6
1.25	0.3	0.3	-0.35	-0.3
1.5	0	0	-0.15	-0.15
1.75	-0.3	-0.25	0	0.05
2	-0.75	-0.6	0.2	0.17
2.25	-1	-0.9	0.35	0.4
2.5	-0.85	-0.8	0.3	0.27
2.75	-0.85	-0.8	0.25	0.25
3	-0.7	-0.65	0.25	0.2
3.25	-0.5	-0.6	0.2	0.2

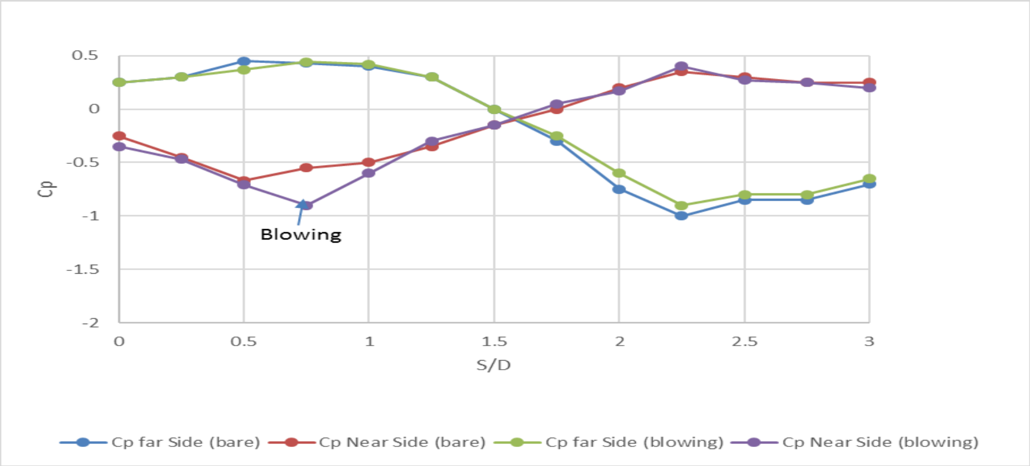

Figure 10 Surface pressure distribution on the side wall at *Re*=1.47x10^4

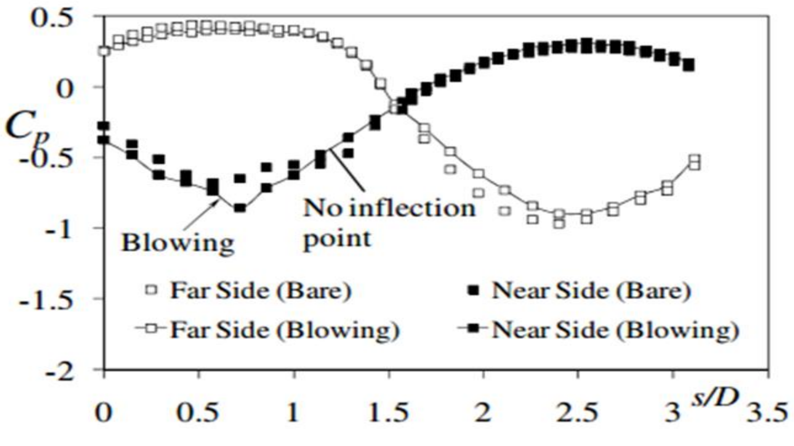

Figure 11. Graph of Cp v/s S/D obtained practically by Thye, N. Y [13] at *Re*=1.47x10^5

From Figure 8 and Figure 10 it is seen that there is no point of inflection in case of duct with Tangential Blowing as compare to bare duct at Re=1.47 X 10^5 so it can be concluded from above chart that flow separation is suppressed by use of flow control device i.e. Tangential Blowing and the pattern obtained by simulation is like the graph of Cp vs S/D obtained practically by Thye, N.Y. i.e. Fig. 11.

Total pressure loss

Table 3. Total pressure loss coefficient with respect to Reynolds number for bare and Tangential Blowing duct.

S. No.	Total pressure loss coefficient	Reynolds number 4.73x10^4		Reynolds number 1.47x10^5	
		Simulation	Experimental [14]	Simulation	Experimental [14]
1.	Bare Duct	0.2	0.24	0.25	0.2
2.	VG	0.115	0.2	0.12	0.13
3.	Tangential Blowing Cm=0.12	0.15	0.16	0.15	0.14
4.	Tangential blowing Cm=0.045	0.12	0.13	0.125	0.12

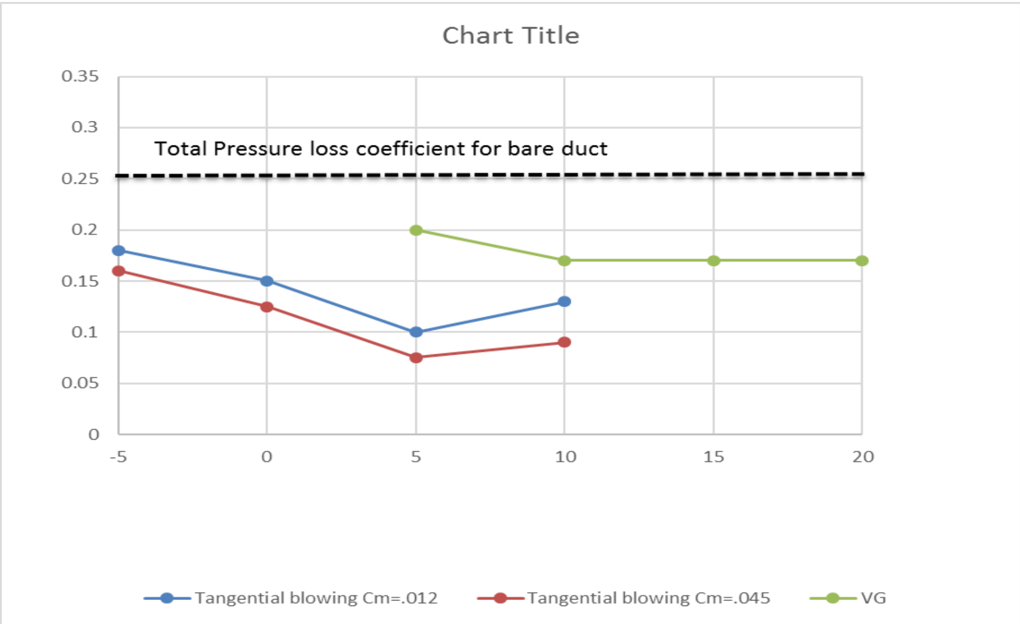

Figure 12. Variation of total pressure loss coefficient at different incident angle for VG and Tangential blowing configuration at Re = 4.73x10^4

Figure 13. Variation of total pressure loss coefficient at different incident angle for VG and Tangential blowing configuration at Re = 1.47×10^5

From Figure and from Figure 13, it is found that there is reduction in total pressure loss coefficient by using vortex generator and tangential blowing. As total loss coefficient is 0.25 and 0.2 for the bare duct respectively for both Reynolds number.

- In case of Tangential Blowing it is 0.115 For Re=4.73×10^4.
- For Reynolds number 1.473×10^5 for Tangential Blowing it is 0.2.

III. CONCLUSION

The optimum aerodynamic performance of S-shaped ducts (or aircraft air-intake ducts) demands that a relatively uniform flow with a smallest possible pressure loss. The contour of static pressure obtained from simulation of bare Duct, shows two low pressure zones are developed on the duct, one at near side wall of the 1st bend and other on the far side wall of the second bend. There is a flow separation in these two pockets of the bend which is clearly seen on the contour of velocity vector (Figure 5). As we use two Reynolds number the effect is same, only difference in magnitude observed. Flow control techniques namely Tangential Blowing is used to suppress the flow separation there why reducing total pressure loss.

The effectiveness of Tangential Blowing to control flow separation, reduce total pressure loss is studied. The graph of Cp vs S/D (Figure 11) of simulation shows no point of inflection i.e. there is no flow separation on these walls. There is also reduction in total pressure loss coefficient by using Tangential Blowing shown in table 3. As total loss coefficient is 0.25 and 0.2 for the bare duct for respective Reynolds number. In the case of Tangential Blowing it is 0.115 and 0.2 for respective Reynolds number.

The method Tangential Blowing was found to be effective in suppressing flow separation and reducing total pressure loss. Therefore, the results indicate competing parameters for improving the performance of flow in S-ducts.

Conflict of Interest: The authors declare that they have no conflict of interest.

Ethical Statement: The authors declare that they have followed ethical responsibilities.

REFERENCES

[1] Anderson, B.H., and Gibb, J., Vortex-Generator Installation Studies on Steady-State and Dynamic Distortion, Journal of Aircraft, Vol. 35(4) Jul-Aug (1998), pp. 513-520.

[2] Anderson, B.H., Reddy, D.R. and Kapoor, K., Study on Computing Separating Flows Within a Diffusing Inlet S-duct, Journal of Propulsion and Power, Vol. 10(5) (1994), pp. 661-667

[3] Anderson, B.H., Taylor, A.M.K.P., Whitelaw, J.H. and Yianneskis, M., 1982. Developing Flow in S shaped Ducts. In: Proceeding of Second Symposium on the Application of LDA to Fluid Mechanics, Lisbon, Portugal, Paper 4.2, 17 p.

[4] Bansod, P., and Bradshaw, P., The Flow in S-Shaped Ducts, Aeronautical Quarterly, Vol. 23 (1972), pp. 131-140.

[5] Cheng, K.C. and Shi, L. Visualizations of Developing Secondary Flow and Measurements of Velocity Profiles in a Curved Square Duct with and without an Offset Bend. In: Flucome '91, 3rd Triennial International Symposium on Fluid Control, Measurement, and Visualization, San Francisco, USA, (1991) pp. 415-422.

[6] Gad-el Hak, M. and Bushnell, D.M., Separation Control: Review, Transactions of ASME: Journal of Fluid Engineering, Vol. 113(1) Mar (1991), pp 5-30

[7] Guo R.W. and Seddon J., 1983. The Swirl in an S-duct of Typical Air Intake Proportions, Aeronautical Quarterly, 34(2) 99-129.

[8] Kitchen, B.J. and Bowyer, J.M. Jr., Towards the Optimization of a Non-diffusing Two Dimensional, S-shaped Duct, In: Forum of Turbulent Flows, ASME Fluid Engineering Division FED, San Diego, USA, Vol. 76 (1989), pp 85-92.

[9] M. Norouzia, M.H. Kayhani, C.Shu, M.R.H. Nobari et al. (2010), Flow of second-order fluid in a curved duct with square cross-section, J. Non-Newtonian Fluid Mech.165 (2010) 323–339.

[10] Rojars, J., Whitelaw, J.H. and Yianneskis, M., Developing Flows in S-shaped Diffusers Part I: Square to Rectangular Cross Section Diffuser, NASA Contractor Report 3631, (1983), 50 p.

[11] Sullerey, R.K., and Pradeep, A.M., Effectiveness of Flow Control Devices on S-Duct Diffuser Performance in the Presence of Inflow Distortion, International Journal of Turbo and Jet-Engines, Vol. 19(4) (2002), pp. 259-270.

[12] Taylor A.M.K.P., Whitelaw J.H. and Yianneskis M., 1984. Developing Flow in S-shaped Ducts 2: Circular Cross-section Duct, NASA Contractor Report 3759, 60 p.

[13] Thye, N. Y., 2009.A Thesis, Doctor of Philosophy, National University of Singapore

[14] Tilak T. Chandratilleke, NimaNadim, Ramesh Narayanaswamy et al (2012) Vortex structure-based analysis of laminar flow behavior and thermal characteristics in curved ducts, International Journal of Thermal Sciences 59 (2012) 75e86.

[15] Whitelaw, J.H. and Yu, S.C.M., 1993a. Turbulent Flow Characteristics in an S-shaped Diffusing Duct, Flow Measurement and Instrumentation, 4(3)171-179.

[16] Whitelaw, J.H. and Yu, S.C.M., 1993b. Velocity Measurements in an S-shaped Diffusing Duct, Experiments in Fluids, 15(3-4) 364-367.

[17] Xiaoyun Wu, Sangding Lai, Kyoji Yamamoto, Shinichiro Yanase et al (2011), Vortex Patterns of the Flow in a Curved Duct, ICSGCE 2011: 27–30 September 2011, Chengdu, China.

[18] Shams, M. N., Singh, R. K., & Zunaid, M. (2016). CFD Modeling of flow through S-Shaped Duct.

