

CFD Analysis of Serpentine Inlet Duct to Enhance the Flow Properties Using Vortex Generator

Santosh Hosur¹, Dr. Basawaraj²

PG Student, Department of Aerospace Propulsion Technology, VTU, CPGS, VIAT, Muddenahalli, Chickaballapura, India¹

Associate Professor, Department of Aerospace Propulsion Technology VTU, CPGS, VIAT, Muddenahalli, Chickaballapura, India²

ABSTRACT: In the current work, computational study has been carried out on the both baseline serpentine duct to know the flow behaviour and baseline with vortex generator to the effect of vortex generators on the flow properties of the duct. A set of vortex generators are chosen for the study, by keeping all other parameters constant. In current work, for modelling the duct and vortex generator CATIA V5 is used; ICEM CFD for discretizing the geometry. For the analysis, the computational fluid dynamics tool ANSYS FLUENT is used, SST k- ω model available in the FLUENT is used for the computations, the results of current work are validated with experimental results conducted by an author kirk and results of present work shows that they are converged. The use of vortex generator in the current work proved extremely effective in enforcing the active flow control. Results obtained in this work are to study the pressure recovery at the engine face. The vortex generator used in the present work have incidence angle of 16° with the incoming flow, The results ensure that the current work on the baseline model of serpentine inlet duct and model with vortex generator of comparison is well established in case of pressure recovery and decrease in engine face distortion. In the present study significant improvements characterized by an improved pressure recovery of 43.48% and decrease in distortion close to 21.80% are observed over the baseline model of serpentine duct for vortex generator model. This reduction in pressure loss and distortion enhances the engine performance of thrust and also improves the overall performance of the engine.

KEYWORDS: Serpentine duct, Vortex generators, Serpentine inlet, flow separation, total pressure recovery; circular total pressure distortion; synthesis distortion coefficient.

Nomenclature:

AOA -Angle of attack

VG -Vortex Generator

S-duct – Serpentine duct

I. INTRODUCTION

Serpentine inlet duct concept exists from long back but the use of these kinds of inlet ducts can be seen in lesser number of military and commercial aircraft like Boeing 727, F-117 and F-16. In commercial aircraft now a day's these serpentine inlet ducts are using as thrust vectors. This concept is worthwhile to maintain the axis of the aircraft to be aligned with rear mount of the engines. Mainly these serpentine ducts will be used in Unmanned Aerial Vehicles (UAVs), where loss of life is not a big concern and enhancing the stability and performance is the main concern.

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 6, June 2016

Development in technology mainly in defence sector needs reduction in radar cross section leads to develop the inlet serpentine duct because they have shorter length and larger offset along the duct. This technology to higher the thrust to weight ratio in aircraft, the 'S' shaped ducts are very ultra-compact, highly offset inlet duct and also which results reduce the overall length of the aircraft. These type of ducts helps in reduction of radar cross section, but they also may result in reduction of propulsive system stability margin¹. The more compact duct (S-duct) results in increase in the turning inside the duct as a result which initiates secondary flow within the duct. The shape of the duct objects the diffusion and dissipation of the secondary flow and it results in high distortion levels at the inlet section of the engine face. This distortion reduces the stability margin of the compressor and turbine of the engine. While distortion may result in high cycle fatigue, which may leads the aircraft as catastrophic loss, or else it may increase the operation cost of the aircraft and also maintenance.

While taking into consideration there are many design difficulties in these S- shaped inlet ducts. In most of the cases, the flow results from the inlet duct won't be acceptable for the further process. So to get the desired efficiency in the engine operation one should control or correct the amount and quality of the flow entering the engine face. Many demonstrations are available while attempts are made to increase the flow quality within the duct to get uniformity over the flow at exit of the duct; the secondary flow results in the duct have a severe effect on the off-design condition of the aircraft². Understanding of off-design effects in serpentine inlet ducts is difficult and the experiments are going on. It may be necessary to revision the design to ensure the required flow quality in every condition that may faces during flight operations⁴. The design revision includes the off-design conditions like angle of attack and asymmetric distortion.

In the simulation of the inlet duct with and without the vortex generators, it has been proven that the results obtained by the baseline inlet duct with vortex generator showed well defined improvements over the base line duct model for the pressure recovery and distortion values. Due to complex shapes and geometry of the serpentine ducts, there are secondary flows and distortions at the engine face which causes problem to the engine performance, that's why it is necessary to understand the flow structure and the effects of secondary flows on the engine performance. The main intention of the work is to study the flow structure inside the serpentine inlet duct and identify the cause for the secondary flows and came up with any of the passive flow controlling technique to improve the performance by altering the flow structure in the inlet duct. The results of present work showing that good improvement of Cp loss compared with pulsed jet blowing and better decrease in Cp loss compared to steady blowing with less decrease in distortion compared to steady blowing with fluidic actuator.

II. MODEL GEOMETRY AND DISCRETIZATION

The serpentine duct considered for the present work consisting of convoluted geometric features. Serpentine duct geometry and flow physics settings are the unique features made the CFD problem more demanding. Grid generation and turbulent model selection constitute unique challenge for the project. The CAD model is generated using CATIA V5-R20 software as shown in the Figure-1, and the geometric specifications of the duct are tabulated in the Table-1. The inlet of the serpentine duct which is as shown in the Figure-1 has two S-duct features with approximately 45° bend with diffusion exit of elliptical to circular section. The duct has an L/D ratio of 2.5 and consists of a 4:1 aspect ratio (AR), entrance section was bi convex and a length of 0.635 m with the exit diameter of 0.254 m. Serpentine duct with vortex generators also modelled in the CATIA V5, here the vortex generators have placed with an angle of incidence 16 degree, and the lateral spacing of the vortex generator is around 16.92mm.

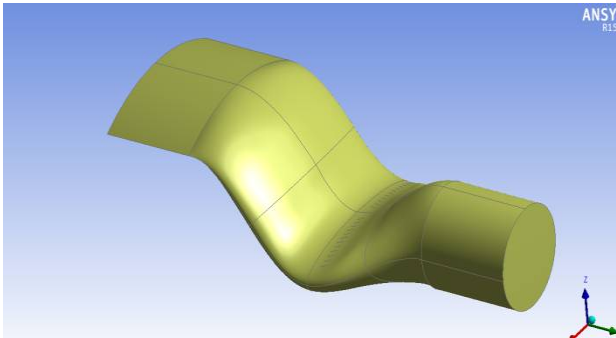


Figure-1 Baseline serpentine duct model

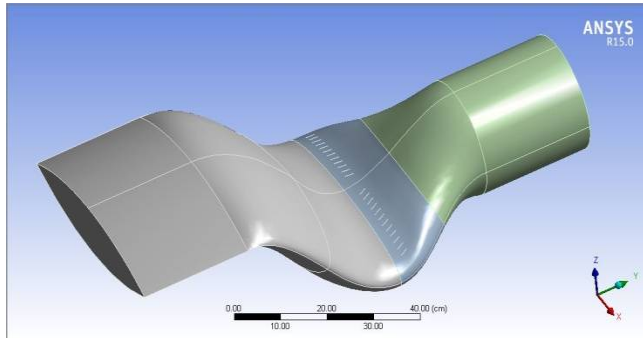


Figure-2 Serpentine duct with vortex generators

In second model (Figure-2), the serpentine duct has the vortex generators in the upper surface of the second bend to nullify the generation of secondary flows. The following Figure-3 shows the CAD geometry of serpentine duct with vortex generator installed.

The below Table-2 shows the dimension specification of the vortex generator. The next step is to export the CAD model to ANSYS ICEM CFD for meshing followed by boundary/solver settings.

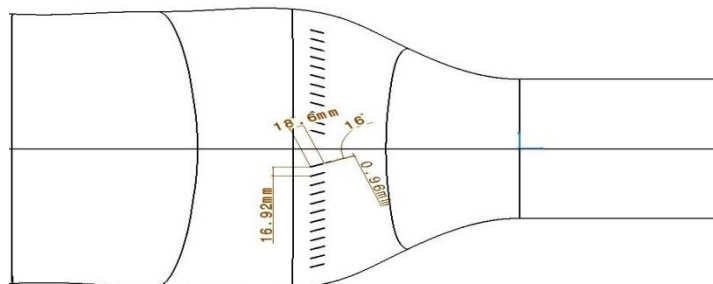


Figure-3 CAD geometry of serpentine duct with vortex generator

Table-2 S-duct with vortex generators details

Specifications	Dimensions
Height	4.86 mm
Length	18.6 mm
Width	0.96 mm
Angle of incidence	16°
Lateral spacing	16.92 mm

Table-1 Baseline duct geometry details

Parameter Specification	Dimensions
Serpentine duct length	0.63 m
Minor diameter of inlet	0.124 m
Major diameter of inlet	0.49 m
Serpentine duct outlet diameter	0.254 m
Length to diameter ratio	2.5 m

III. DISCRETIZATION OF THE MODEL

A three dimensional serpentine duct CAD model has been imported from CATIA V-5 R-20. The discretization of CAD model (serpentine duct) has been done using ANSYS ICEM CFD. The quality of the grid is often an issue which has to be taken into consideration in computational analysis and its consequences greatly influence the results. To understand the flow properties, the initial discretization of serpentine duct has been done with coarse mesh retaining the features of duct geometry. The following Figure-4 shows the finer discretization of the duct model. As it stated earlier flow field properties are grid dependent, hence to study the boundary layer separation and generation of vortices the grid has to refine near to the curved profile (the first and second bend region) as shown in the Figure-4. Grid convergence simply taken to the ability of the grid to minimal interferes with the solution. The simplest test for grid convergence study is the usage of different mesh densities with varying quality starting from lower to higher.

Grid convergence is assumed if there is an equitable agreement between the results from the different mesh densities. The following Table-3 represents the different densities of mesh.

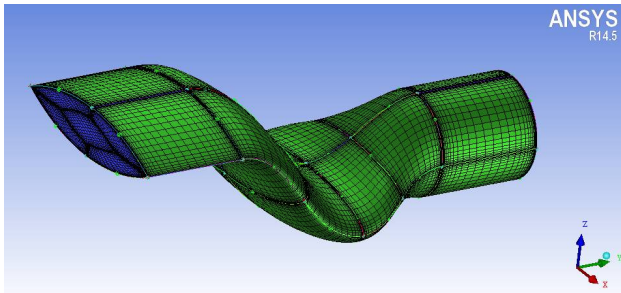


Figure-4 Baseline serpentine duct meshing

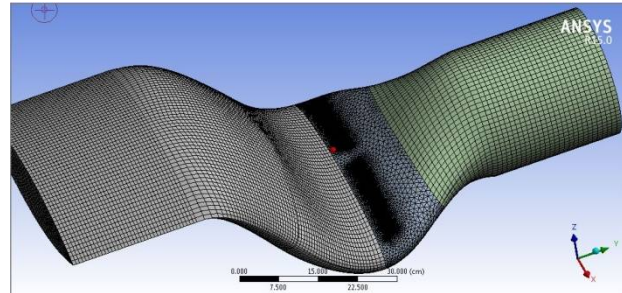


Figure-5 S-duct with vortex generators meshing

The serpentine duct with vortex generator also discretised in the ANSYS WORKBENCH. For discretizing the serpentine duct with VG's is bit difficult, because the vortex generator present in the model needs high accuracy but its time consuming for high density mesh. So for the current work the model is discretised using the hexahedral core mesh at the inlet and outlet, but the vortex generator part is meshed with high density tetrahedron core mesh as shown in the above Figure-5.

Table-3 serpentine duct grid density

S.1	Grid Types	Elements Density	Orthogonal Quality
1	Grid Density I	1056000	0.15
2	Grid Density II	1805076	0.3
3	Grid Density III	2064000	0.339
4	Grid Density IV	2478960	0.310

IV. SOLVER SETTINGS AND BOUNDARY CONDITION

The following solver parameter represents the selection of solver settings for the duct and boundary condition for the serpentine duct analysis General Methods

❖ Orthogonal quality	3.30430e-01
❖ Solver	Pressure-Based
❖ Viscous Model	k- ω SST turbulence model
❖ Material	Fluid (at sea level)
❖ Inlet boundary	Pressure-Inlet
❖ Outlet boundary	Pressure-Outlet
❖ Serpentine duct wall	Wall (No-Slip condition)
❖ Operating pressure	101325 Pa
❖ Pressure-Velocity	Simple scheme
❖ Gradient	Least square based
❖ Pressure	Standard
❖ Momentum	Second ordered Upwind scheme
❖ Turbulent Kinetic Energy	Second ordered Upwind scheme
❖ Specific Dissipation Rate	Second ordered Upwind scheme
❖ Specific Dissipation Rate	0.8
❖ Turbulent Viscosity	1
❖ Initialization Method	Standard Initialization
❖ Mach No	Less than 0.3

V. EXPERIMENTAL RESULTS

Baseline serpentine duct

The present sub-section will consolidate the results of Computational Fluid Dynamic analysis of a baseline serpentine duct, followed by the flow control aerodynamic devices, i.e. vortex generators. The CFD analysis has been carried out for different volume mesh densities and best suitable results has been opted for the problem defined. The following Figure-6 shows the dynamic pressure contour plots for grid independence study.

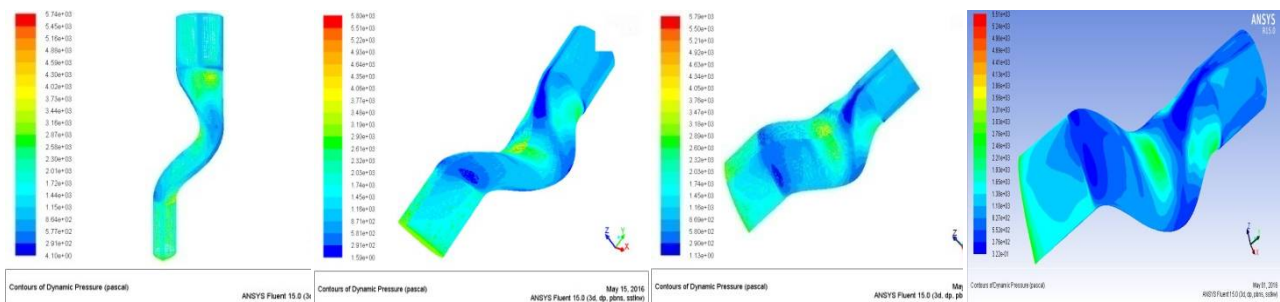


Figure-6 Dynamic pressure contour for baseline ducts

The analysis has done based on tabulated grid densities as shown in the Table-3. To compare the CFD results with references⁴, the distortion and pressure loss value were analysed. The results of grid densities 2034560 and 2478960 does not varies significantly, hence considering the required accuracy of the results and resources the grid of density 2478960 is considered as final grid for the further analysis.

The following Figure-7 represents the variation of total pressure contours at many axial locations in the duct, while Figure-8 shows the group of streamlines, represented by Mach number that indicates formation of vortex at the second bend. By combining the information given by each of these plots, the generation of secondary flow development as discussed in the introductory section of the report is confirmed. At the second bend of the duct flow separation leading to low pressure region near the wall, which are concentrated towards the centre of the duct. These pressure deficiencies cause the flow to rush in from the side of the duct wall, converging at the centre line where the flow is forced into vertical pattern. The baseline duct shows separation of flow in the same area as in reference⁴.

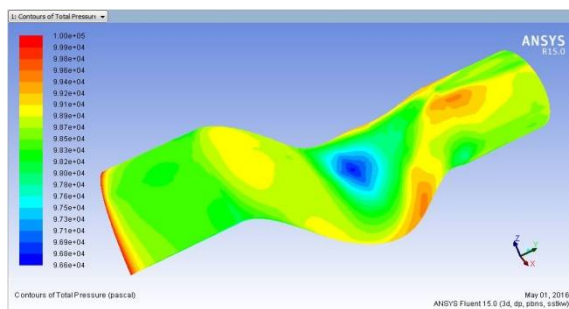


Figure-7 Represents the variation of total pressure contour

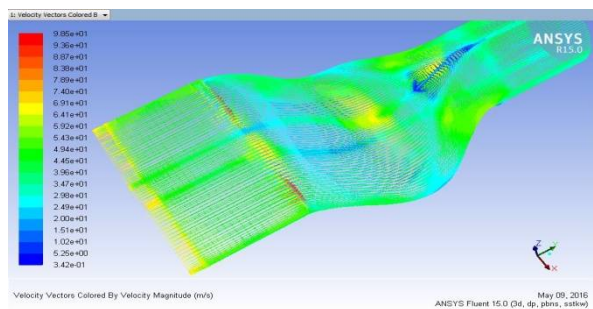


Figure-8 Streamline trace in Fluent

The analysis shows the strong evidence of secondary flow generation and boundary layer formation which are substantial at near wall region as shown in the following velocity vectors (Figure-9). Reverse vector clearly indicating the diffusion of flow at the main stream (first and second bend of the duct).

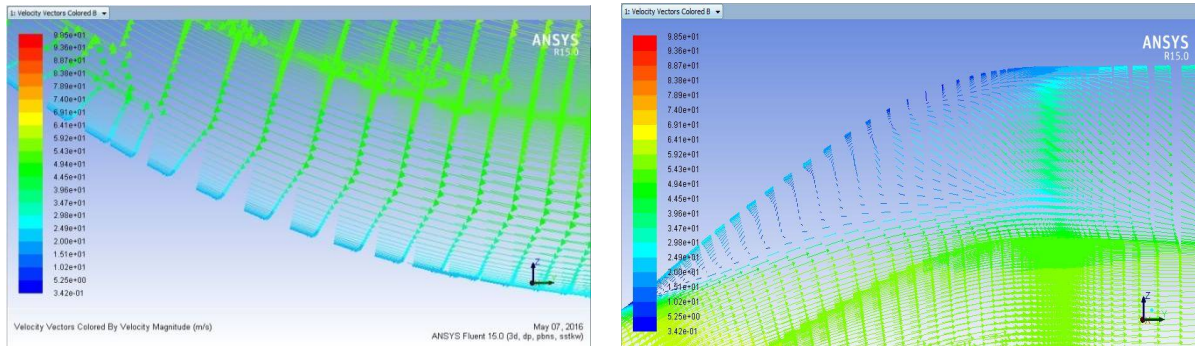
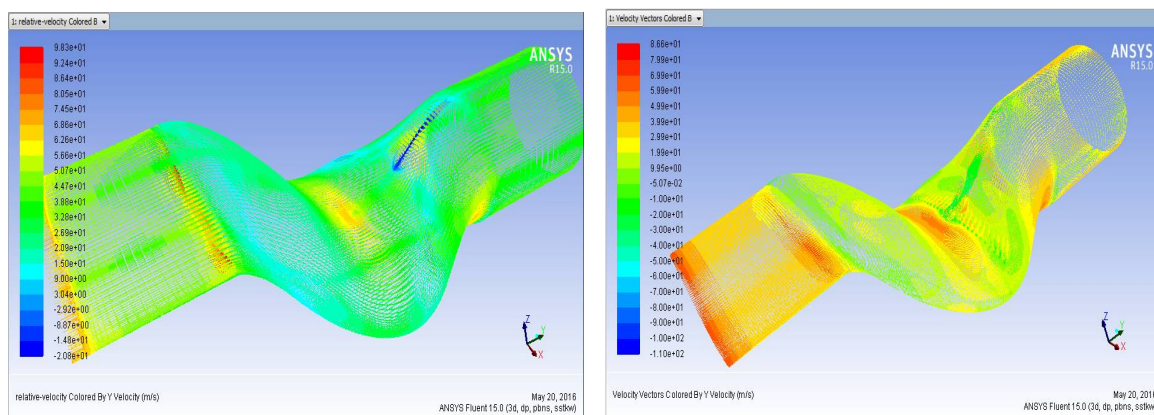


Figure-9 Separation at the bends of baseline duct

The secondary flow at the bends section continues to exit and circulate well into the engine face without any cancellation effects. This is an important perception in the understanding of secondary flows in S shaped ducts proposing the irreversible nature of boundary layer migration; it continues to migrate even after the first bend. Finally at the bottom surface of the duct irreversibility of moment is verified by continuation of flow in the centre gap, an after effect of first duct bend. The flow separation finally ends up in the rotating vortex pairs at the inlet of the engine face.

Effect of Vortex Generators

The numerical analyses were run for the same boundary conditions as of the baseline simulations the results came in 1200 iterations. The analysis results obtained for the duct with vortex generators are compared well with the baseline serpentine duct results to analyse the effects of vortex generators on the flow behaviour. The following Figure-10 shows the comparison of velocity along the Y-direction. With the comparison it can be seen that, there is a flow separation near the second bend of the baseline serpentine duct, resulting in to reverse flow leading to generation of the secondary flow. Whereas duct with vortex generators, there is no reverse flow at the second bend, flow separations are nullified with the introduction of vortex generators.



Baseline duct
Vortex generator
Figure-10 Contours of velocity vectors along the axial direction.

The following Figure-11 compares the vorticity magnitude of without VG duct and duct with vortex generators. It shows that there is formation of vorticity due to installation of vortex generator, the same kind of vorticity found in the Kirk experimental results by providing fluidic actuator with slots.

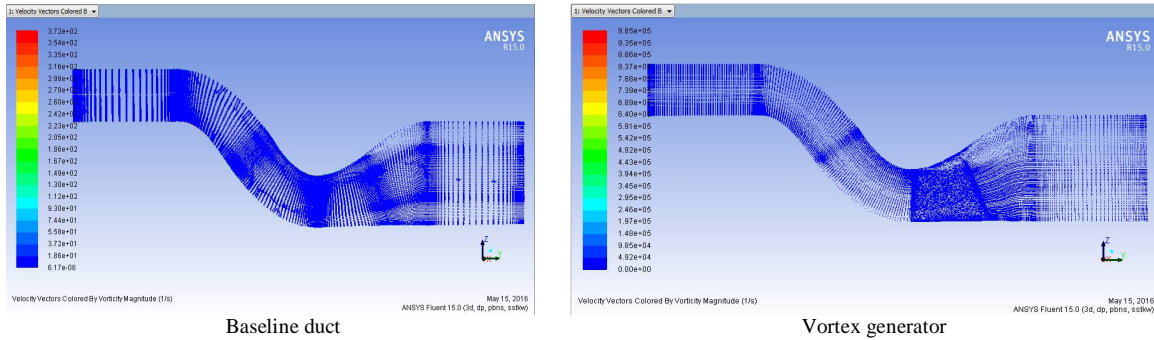


Figure-11 Compares the vorticity magnitude

In the following Figure -12 shows that the variation of the total pressure at the engine face for both baseline duct and vortex generator model, it is clear from the figures that the pressure recovery at the engine face is good in vortex generator model compare to baseline duct model

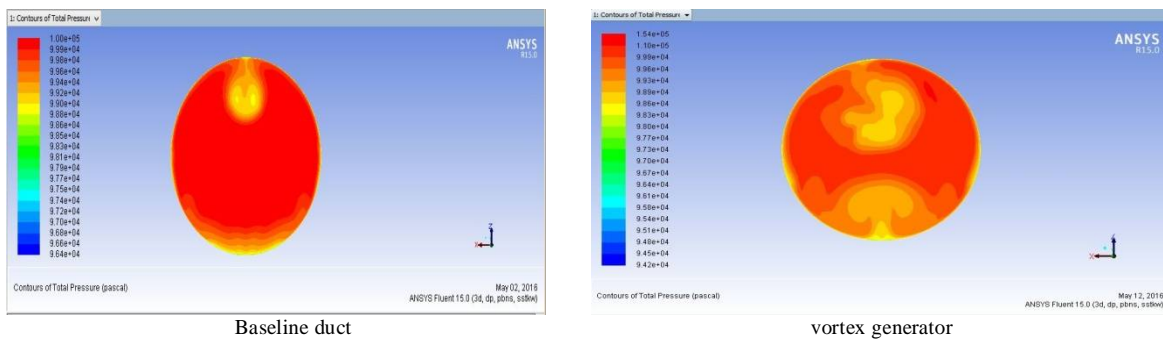


Figure-12 Compares the total pressure at the engine face

In the following validation for the C_p loss and distortion is given in Table-4. It includes the results of experimental and literature to compare with the current work; it shows that we have well agreement between the Kirk experimental results² and Present work. C_p loss and distortions are mentioned in the percentage to provide the good comparison.

From results table shows that by introducing Vortex generators which are controlling the pressure loss at the engine face, here about 43.48% of coefficient pressure (C_p) loss reduction is observed, when compared with baseline duct. Serpentine duct with vortex generators are efficient in controlling the distortion, a 21.80% of reduction in the engine face distortion when compared with baseline duct.

Table-4 Validation of C_p loss and Distortion

	Kirk et al ²					Present work		
	Experimental baseline in (%)	Experimental fluidic actuator (%)	Decrease in loss in (%)	Decrease in loss by steady blowing in (%)	Decrease in loss by pulsed jet in (%)	Baseline duct in (%)	Vortex generator duct in (%)	Decrease in loss in (%)
C_p loss	44.01	20.85	52.62	37	16.5	47.10	26.62	43.48
Distortion	79.38±3.18	70.2	14.97	41	--	72.59	56.76	21.80

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 6, June 2016

VI. CONCLUSION

The following conclusions are made from the present studies.

1. The baseline computational results show the same trend as of the experimental results in the literature. However, there are some differences in the Co-efficient of pressure loss and distortion numbers when compared to experimental results.
2. Secondary flow development has nullified by using vortex generator in the second bend.
3. Vortex generators are much impressive in controlling the pressure loss at the engine face, here about 43.48 % of C_p pressure loss reduction is observed, when compared with baseline.
4. Serpentine duct with vortex generators are efficient in controlling the distortion, a 21.80% of reduction in the engine face distortion
5. It is clear that by installing the vortex generator boundary layer separation will be much lesser by leaving the high energy core flow and by distributing the low energy flow uniformly inside the periphery of the engine face.
6. Pressure distribution at the engine face is relatively good in inlet duct with vortex generator compared to baseline duct. The results of present work showing that good improvement of C_p loss compared with pulsed jet blowing and noticeable decrease in C_p loss compared to steady blowing with less decrease in distortion compared to steady blowing with fluidic actuator.

REFERENCES

- [1] J.W. Hamstra, D.N. Miller, P.P. Truax, B.A. Anderson, and B.J. Wendt. 'Active inlet flow control'; technology Demonstration'. ICAS 2000 CONGRESS, Published by the International Council of the Aeronautical Sciences.
- [2] Aaron Michael Kirk, B.S. 'Active Flow Control in an Advanced Serpentine Jet Engine Inlet Duct'. Texas A&M University, 2006.
- [3] Ritesh Gaur, Vimala Narayanan and Kishore Kumar S. 'Influence of Area Distribution with fixed Trajectory on Serpentine Intake Duct Performance' ASME-2012 Gas Turbine India Conference, GT India 2012-9625
- [4] Kirk, Aaron M., Gargoloff, Joaquin I., Rediniotis, Othon K. and Cizmas, Paul G.A. 'Numerical and experimental investigation of a serpentine inlet duct'. International Journal of Computational Fluid Dynamics, 23: 3, 245— 258, 2009.
- [5] Vakili et al. 'Design and analysis of vortex, generator for a HEV model'. M. Sc.
- [6] Bernhard H. Anderson and James Gibb. 'Vortex Generator Installation; Studies on Steady State and Dynamic Inlet Distortion'. 32nd Joint Propulsion Conference cosponsored by AIAA, ASME, SAE, and ASEE Lake Buena Vista, Florida, 1996.
- [7] Clara Marika Velte. 'Characterization of Vortex,, Generator Induced Flow'. Fluid Mechanics Department of Mechanical Engineering Technical University of Denmark, July 2009.
- [8] Abhinav Kumar.. 'Flow Control Optimization in a Jet Engine Serpentine Inlet Duct'. Office of Graduate Studies of Texas A&M University, August 2007.
- [9] Reichert, B. A. and Wendt, B. J., "Improving Diffusing S-Duct Performance by Secondary Flow Control," AIAA Paper 94-0365, January 1994.
- [10] ANDERSON, BERNHARD, and JAMES GIBB. "Application of computational fluid dynamics to the study of vortex flow control for the management of inlet distortion", 28th Joint Propulsion Conference and Exhibit, 1992. Publication Dudek, Julianne.
- [11] C M Velte, M O L Hansen and D Cavar. 'Flow analysis of vortex generators on wing sections by stereoscopic particle image velocimetry measurements'. Technical University of Denmark, Department. of Mechanical.. Engineering, Fluid Mechanics Section, 2800 Kongens Lyngby, Denmark, 24 January 2008.
- [12] Spalart, P. R., Allmaras, S. R.: 'A. One-equation Turbulence Model for Aerodynamic Flows'. AIAA, Paper 92-0439, 1992.
- [13] Dr. Sonya T. Smith, Dr. Angela. Scribden and Matthew Goettke. 'Computational Study of Inlet Active Flow Control'. Howard University Department of Mechanical Engineering Washington, DC. MAY 2007.
- [14] Vincent, Robert, and Raymond Mapple. "CFD Investigation of Laminar;" Flow Over a Dimpled Surface Indentation", 36th AIAA Fluid Dynamics Conference and Exhibit, 2006.
- [15] Dudek, Julianne. "An Empirical Model for Vane-Type Vortex Generators in a Navier-Stokes Code", 43rd AIAA Aerospace Sciences Meeting and Exhibit, 2005
- [16] Poisson-Quinton, Ph. and Lepage, L., "Survey of French Research on the Control of Boundary Layer and Circulation," Boundary Layer and Flow Control, Vol. Pergamon Press, New York, 1961.