70(10A): 324-330; 2022. DOI: 10.18311/jmmf/2022/32776

Print ISSN : 0022-2755

Journal of Mines, Metals and Fuels

Contents available at: www.informaticsjournals.com/index.php/jmmf

Design and Numerical Study of Serpentine Duct using Flow Controllers for Flow Characteristics Enhancement

Yogesh Shankar R.P[a*] and Pavan D[b]

[a]*School of Mechanical Engineering, REVA University, Bangalore 560064,Karnataka, India.*
[*]*Email: yogeshshankar003rp@gmail.com*
[b]*Assistant Professor, School of Mechanical Engineering, REVA University, Bangalore*

Abstract

In the current work, computational study has been carrying out on the both baseline serpentine duct to know the flow behaviour and baseline with vortex generator to know 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. Here in current work, for modelling the duct and vortex generator CATIA V5 is used; to discretizing the geometry ICEM CFD is used. For the analysis, the computational fluid dynamics tool FLUENT is used, SST k-ù model available in the FLUENT is used for the computations, and the results of current work will be validated with experimental results and shows that they are converged. 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 results ensure that the current work on the baseline model of serpentine inlet duct and model with vortex generator comparison is well established in case of pressure recovery and decrease in engine face distortion. After comparing the result of both baseline duct and duct with vortex generator we are expecting at least 25-35% of pressure recovery at the engine face. In the present study significant improvements characterized by an improved pressure recovery and decrease in distortion 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: *CATIA V5, ANSYS workbench software, FLUENT, ICEM CFD, SST k-ù*

1.0 Introduction

Serpentine intake ducts have been around for a long time, although they are only seen in a few number of military and commercial aircraft, such as the Boeing 727, F-117, and F-16. These serpentine inlet ducts are now used as thrust vectors in commercial aircraft. This approach is worthwhile in order to keep the aircraft's axis aligned with the engine's rear mount. These serpentine ducts will mostly be employed in Unmanned Aerial Vehicles (UAVs), where loss of life is less of a worry than improving stability and performance. Because inlet serpentine dust has a shorter length and a larger offset along the duct, it has been developed as a result of technological advancements, particularly in the defence sector.

The 'S' shaped ducts are very ultra-compact, highly offset inlet ducts, and minimize the total length of the aircraft as a result of this technology to improve the thrust to weight ratio in aircraft. These types of ducts aid in the decrease of radar

*Corresponding Author

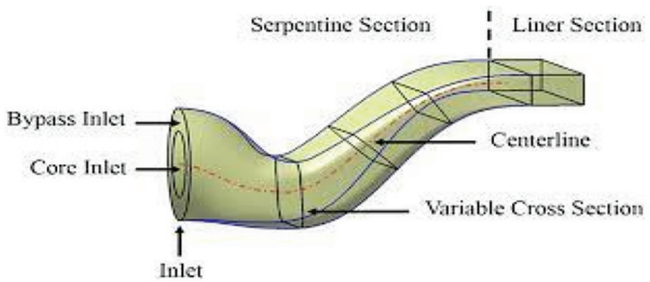

Figure 1: Representation of S-Duct

cross section, but they may also reduce the stability margin of the propelling system. As a result of the increased twisting inside the duct (S-duct), secondary flow is initiated within the duct. The duct's geometry prevents secondary flow diffusion and dissipation, resulting in significant distortion levels at the engine face's inlet portion. The compressor and turbine of the engine's stability margins are reduced as a result of this distortion. While distortion may cause severe cycle fatigue, resulting in catastrophic loss of the aircraft, or it may raise the aircraft's operation and maintenance costs.

There are numerous design challenges in these S-shaped inlet ducts to be considered. In most circumstances, the flow results from the inlet duct will be insufficient for the next step in the process. To achieve the necessary efficiency in engine operation, the amount and quality of the flow entering the engine face must be controlled or corrected. Many demonstrations are available, and efforts are being made to improve the flow quality within the duct in order to achieve consistency in the flow at the duct's outlet; in the literature review, these will be discussed. The duct's secondary flow has a significant impact on the aircraft's off-deign condition. Off-design effects in serpentine inlet ducts are challenging to understand, and investigations are ongoing. It may be necessary to make changes to the design to ensure that the requisite flow quality is maintained in all conditions encountered during flight operations. Off-design conditions such as angle of attack and asymmetric distortion are included in the design revision.

Besides of the advantages like highly offset, and diffusing, ultra compact S-ducts off design conditions face the significant design difficulties. AIP (aerodynamic interface plane) faces the distortion creates by the inlet serpentine duct, where the engine faces meets the exit of the S-duct, where the engine faces followed by the propulsion system of the aircraft. Engine face distortions may rise to unacceptable levels in terms of high cycle fatigue and/or stability margin due to flight conditions such as angle of attack. This research is brought to increase the capacity of static ground testing by using different flight conditions for increasing the ability to simulate. The present work is to develop a method to find the effect of off-design conditions like angle of attack and asymmetric inlet distortion. Many researchers carried out their work on S-duct using fluid actuators for example Kirk had studied for the fluid actuators with steady state blowing and pulsed jet blowing and observed that comparative decrease in Cp loss and distortion with baseline duct. And the present study is carried for vortex generator at 16° of AOA to examine the effect of flow properties

Here the way to flight design can be examined by ground testing and advanced analysis tools at beginning design step. Generally wind tunnel is used to test these off-design flight conditions following the static ground test phase. It is very expensive and needs more resources as the facility, where in the wind tunnel testing it includes fore body and other pertinent aircraft structures. So it is very difficult to change the design at this stage and moreover expensive also. This can be removed by pre-testing off design flight conditions such as angle of attack, in the static ground test level.

Figure 2: Serpentine duct with natural vortices

Lockheed Martin develops the serpentine duct and flow control techniques. For angle of attack, cruise condition and/ or asymmetric distortion flight cases the experimental test was conducted to find the change in inlet flow control parameters and inlet performance for different flight cruise conditions. To determine the characteristics of flow from the inlet duct at varying conditions the flow at the AIP (Aerodynamic interface plane) has examined in full extent.

Kirk carried out the work for the fluid actuators by keeping the slots at second bend to simulate and observe the flow properties and obtained the results, which were of Cp loss of 20.85 % and distortion value of 70.2%. Now the current work is to observe the flow properties using vortex generator (a rectangular strip) at 16°AOA instead of slots for observation of properties like Cp loss and distortion values. The results produced by the baseline intake duct with vortex generators demonstrated well defined improvements over the base line duct model for pressure recovery and distortion values in simulations of the inlet duct with and without vortex generators. The baseline duct has a Cp loss of 47.10 per cent and a distortion of 72.59 per cent, whereas the vortex generator has a Cp loss of 26.62 per cent and a distortion of 56.76 per cent, according to the analysis. By comparing the results of duct with vortex generator over the baseline duct, the results clearly shows that there is a decrease in the Cp loss of 43.48% and distortion decreased of 21.80%.

The results of present work showing that good improvement of Cp loss compared to 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.

2.0 Problem Statement

Secondary flows and distortions at the engine face are caused by the serpentine ducts' complicated forms and geometry, causing problems with engine performance. It is therefore vital to understand the flow structure and the consequences of secondary flows on engine performance. The major goal of this project is to investigate the flow structure inside the serpentine inlet duct, determine the origin of secondary flows, and devise a passive flow control strategy to increase performance by adjusting the flow structure in the inlet duct.

2.1 Problem Statement

CFD analysis of a serpentine inlet duct using a vortex generator to improve flow properties

2.2 Project Objective

- To understand the flow strucure inside the duct as well as to obtain the geometrical details from the review of the literature.
- To descritise the the duct and carry out CFD analysis on the baseline duct and to validate the results with the results obtained from the literature.
- By reviewing the litetratures on the different flow control techniques used, impliment one of them to improve the flow characteristics.
- To study the effect of the vortex gemerator employd in the serpentine duct forthe flow characteristics.
- To carry out the basic study for the different specifications of the vortex generators and to compare the results with the baseline results.
- To justify the changes in flow characteristics and to compare the performance of the serpentine inlet duct with and without vortex generator.

3.0 Methodology

- The geometry was created using commercially available CAD software CATIA V5 R19.
- The geometry was discretized by using ANSYS 15.0 ICEMCFD.
- The flow simulation was done using ANSYS 15.0 FLUENT.
- The boundary condition for the simulation was obtained from the literature review.
- The baseline result was validated with the results obtained from the literature review.
- The vortex generator was used in the inlet duct at the position based on the flow separation.
- The effect of vortex generators on the flow properties and on the flow characteristics was studied
- The vortex generator with different parameters was incorporated and its effects were analyzed.

4.0 Model and Set up Procedure

The serpentine duct under consideration for this study has complex geometric features. The unique features of the serpentine duct design and flow physics conditions made the CFD challenge more difficult. The project has a unique difficulty in terms of grid generation and turbulent model selection. Similar geometries 1, 2 have been modelled for the project in question, however these issues are extremely sensitive to geometric modifications. The problem, on the other hand, has been tackled in a stepwise and systematic manner. The serpentine duct geometry is depicted in Fig.3.

The baseline model of serpentine duct considered is to carry out a systematic study and observe the developed secondary flow and its behaviour how to prevent them, to

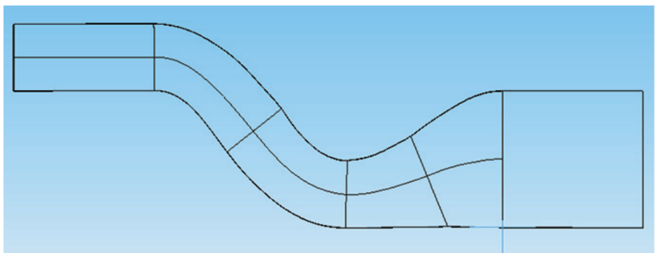

Figure 3: Serpentine duct geometry

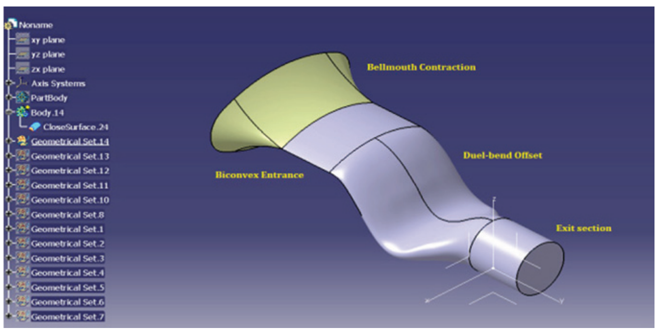

Figure 4: Geometry of the Serpentine Inlet Model

Table 1: Geometric details of the Serpentine duct

	Parameter Specification	Dimensions
1.	Serpentine duct length	0.63 M
2.	Minor diameter of inlet	0.124 M
3.	Major diameter of inlet	0.49 M
4.	Serpentine duct outlet diameter	0.254 M
5.	Length to diameter ratio	2.5 M
6.	First bend from inlet	0.254 M
7.	Second bend from inlet	0.35 M

achieve this S type duct has been modelled. Complete CAD geometry of the duct as shown in Figure 2. As illustrated in Figure 4, the CAD model is created with CATIA V5-R20 software, and the duct's geometric parameters are listed in Table 5.1.2 (a) An ultra-small serpentine inlet duct designed and built by Lockheed Martin Aeronautics Company [2,3] is the relative flow channel of interest in this research. The serpentine duct's entrance, as illustrated in Figure 4, includes two S-duct features, one with a 45^o bend and the other with a diffusion exit from an elliptical to a circular section. The duct features a 2.5 L/D ratio, a 4:1 aspect ratio (AR), a bi-convex entry section, and a length of 0.635 m with an exit diameter of 0.254 m.

5.0 Vortex Model Geometry

In this part the serpentine duct with vortex generators model is being designed using CATIA V5 software. The duct geometry specifications are as same as in the baseline duct geometry, but the vortex generator specifications are new in this duct. Where bellow figure shows the S-duct with vortex generator model. The vortex generators are located on the upper surface of the second curve in this model of the serpentine duct. Vortex generators have some required dimensions in this modelling, as determined by the literature. Here are the duct geometry and vortex generator specs. The vortex generators are angled at 16 degrees of incidence, and the lateral spacing of the vortex generators is roughly 16.92 mm. The dimensions of the vortex generator are listed in the Table 2.

6.0 Discretization of The Model

CATIA V-5 R-20 was used to import a three-dimensional surpentine duct CAD model. The ANSYS ICEM CFD was used to discretize the CAD model (serpentine duct). The quality of the grid is frequently an issue that must be considered in computational analysis, and its repercussions have a significant impact on the results. To better understand the flow properties, the initial discretization of the surpentine duct was done with a coarse mesh while keeping the duct geometry in mind. The initial dicretization of the duct model is shown in Figure 6. As previously stated, flow field attributes are grid dependent; so, to analyse boundary layer separation and vortex production, the grid must be refined near the curved profile (the first and second bend region) as shown in the Figure 6. Grid convergence refers to the grid's capacity to minimise interference with the solution. The most basic grid convergence test is to use several mesh densities with variable quality levels, going from low to high. Grid convergence is assumed if the findings from the various mesh densities are in reasonable agreement. The different mesh densities are represented in Table 3. Four separate meshes

Table 2: Vortex generator specifications

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

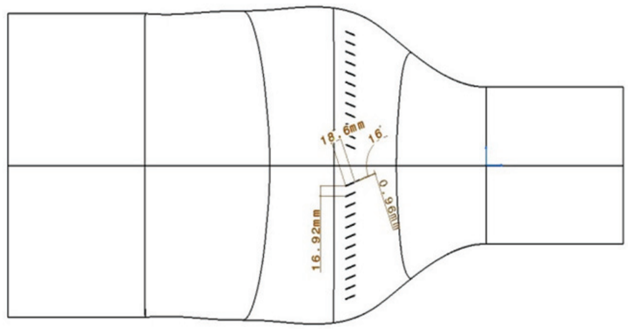

Figure 5: Vortex generator specifications

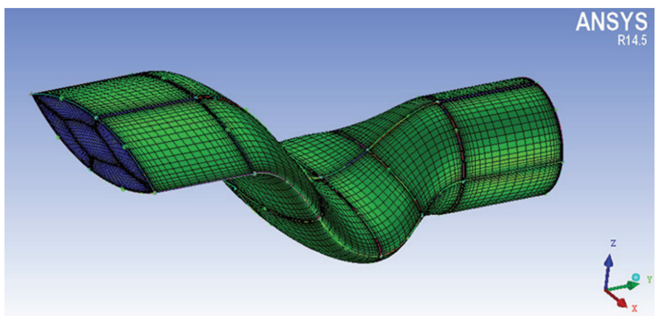

Figure 6: Discretization of surpentine duct baseline geometry

Table 3: Grid types and density

	Grid types	Elements density
1.	Density of the Grid I	1056000
2.	Density of the Grid II	1805076
3.	Density of the Grid III	2064000
4.	Density of the Grid IV	2478960

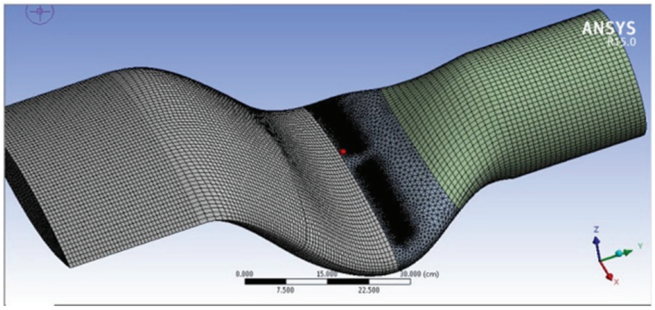

Figure 7: Descretized serpentine duct with Vortex generatoor

with varied mesh densities are created for the surpentine duct geometry. As shown in Figure 6, the node density of various grids varies with different ratios, meaning that the medium quality grid has refined mesh density and that the fine grid has more refined grids than the coarse grid.

7.0 Results and Discussions

The results of a baseline serpentine duct's Computational Fluid Dynamic analysis will be consolidated in this part, followed by flow control aerodynamic devices, such as vortex generators. A three-dimensional unstructured mesh with Reynolds Averaged Navier-Stroke and k- ù turbulence model was used in the numerical analysis for the problem. The project's goal is to understand the physics that govern the flow variables by studying the behaviour of the flow field through the baseline inflow duct and flow control device.

The fluctuation in total pressure contours at several axial places in the duct is depicted in Figure 9 whereas Figure 10 depicts a group of streamlines, represented by Mach number, that indicates the creation of a vortex at the second bend. The genesis of secondary flow development, as stated in the report's introductory section, is confirmed by merging the information provided by each of these charts. At the duct's second bend, flow separation occurs, resulting in a low-pressure zone along the wall that is focused towards the duct's center. Because of the pressure differences, the flow rushes in from the side of the duct wall, converges at the center line, and is forced into a vertical pattern.

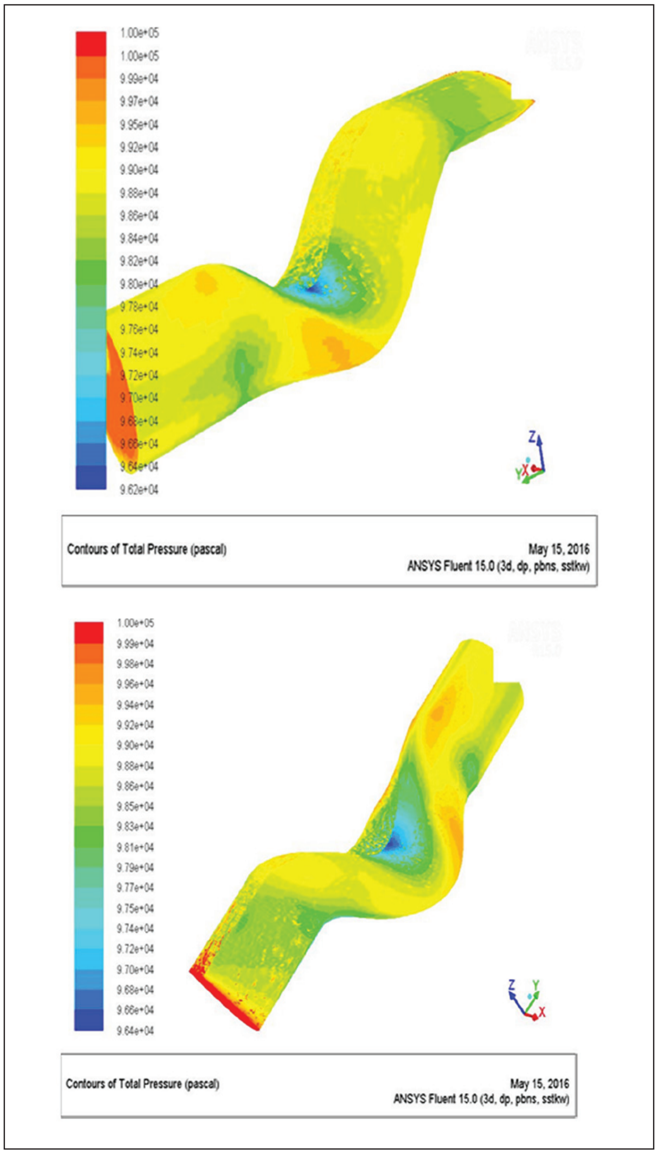

Figure 8: Total Pressure contours for baseline ducts

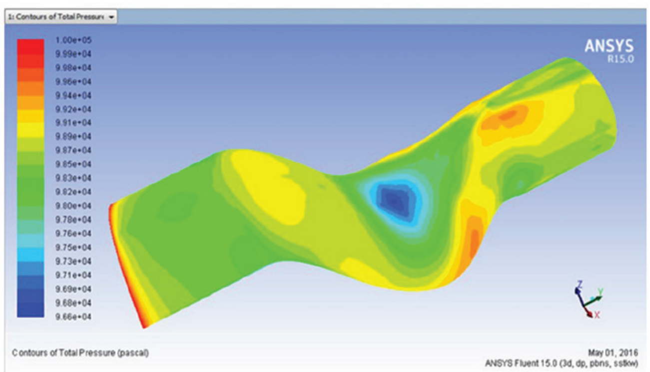

Figure 9: Depicts the total pressure contour fluctuation.

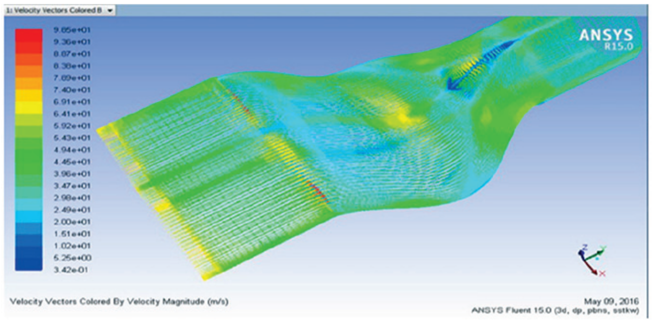

Figure 10: Streamline trace in Fluent

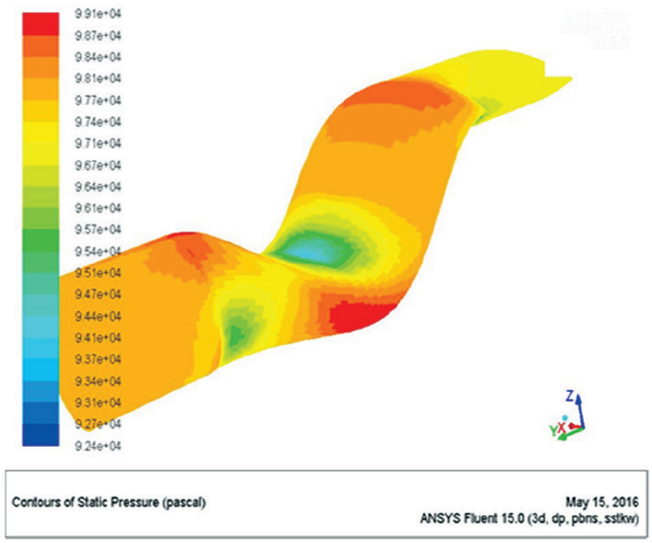

Figure 11: Static pressure contours for baseline ducts

Figure 12 shows the total pressure variation, the green layers close to the wall are because of the viscous diffusion resulting in the formation of boundary layer. At the downstream of the duct flow separation and formation of secondary flow leads to heavy eddy formation, turbulent mixing and flow retardation.

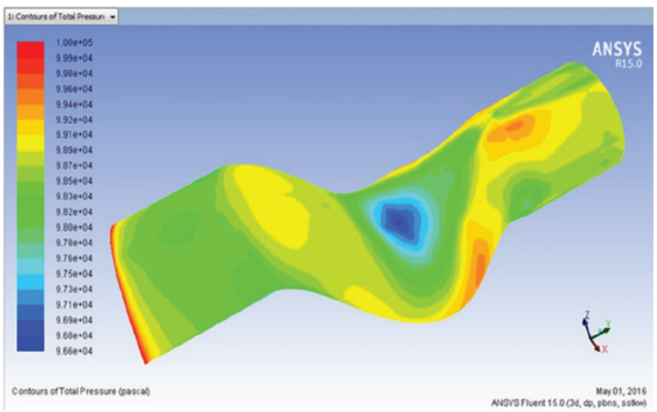

Figure 12: Total pressure contour for baseline duct

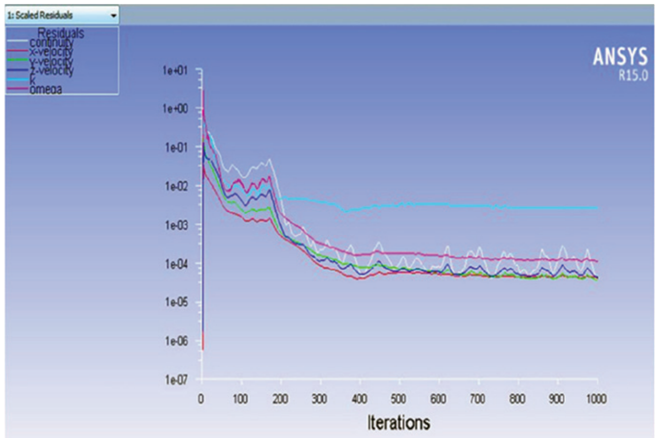

Figure 13: Convergence plot for current work

Figure 13 shows that convergence of a graph for the analysis, from 500 iterations and it is found that analysis results are converged and for the further iterations there was not much change in the residual values hence considering the present iterations for convergence criterion. Figure b shows the convergence graph for the literature, both the graphs shows that the results are satisfactorily converged and the present computation gives good agreement with the literature work.

7.0 Conclusions and Future Work

7.1 Conclusion

From the literature the vortex generators were selected within the duct to know how will be its effect on the flow properties such as distortion and Cp loss. The results were laid on understsnding the flow characteristics with the vortex generators incorporated and changes in coefficient of pressure loss and distortion at the engine face. 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) Vane type vortex generators are effective in preventing the generation of secondary flow.
(4) Vortex generators are much impressive in controlling the pressure loss at the engine face, here about 47.72% of Cp pressure loss reduction is observed, when compared to baseline.

(5) Serpentine duct with vortex generators are efficient in controlling the distortion, a 21.80% of reduction in the engine face distortion.

(6) It is clear that by installing the vortex generator boundary layer seperation 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.

(7) Pressure distribution at the engine face is ralatively good in inlet duct with vortex generator compared to baseline duct.

(8) The current study's findings reveal a significant reduction in Cp loss when compared to pulsed jet blowing and a better drop in Cp loss when compared to steady blowing, with a lower decrease in distortion when compared to steady blowing with a fluidic actuator.

7.2 Recommendations for Future Work

1. The effect different heights and chordlengths of the vortex generator can be analysed in order to get an optimal combination.
2. Installation of axial located vortex generators and its effects need to be studied as well.
3. For different flight conditions like different angles of attack and altitude simulations need to be done.
4. Using geometric optic approximation, effect of these flow control devices on radar cross section can be studied.

Reference

[1] Numerical and Experimental Investigation of a Serpentine Duct Authors: Kirk Aaron M, Garfoloff, Joaquin I, Rediniotis, Othon K, Cizmas, and Paul G A *International journal of Computational Fluid Dynamics*, 23: 3, 245-258; 2009. https://doi.org/10.1080/10618560902835558

[2] Influence of Area Distribution with Fixed Trajectory on Serpentine Intake Duct Performance Authors: Ritesh Gaur, Vimala Narayanan and Kishore Kumar S ASME-012 Gas Turbine India Conference, GT India 2012-9625 https://doi.org/10.1115/GTINDIA2012-9625

[3] Active Flow Control in an Advanced Serpentine Jet Engine Inlet Duct Author: Aaron Micheal Kirk. Texas A&M University, 2006. https://hdl.handle.net/1969.1/ETD-TAMU1027

[4] Vortex-Generator Installation Studies on Steady-State and Dynamic Distortion Authors: Bernard H. Anderson (NASA Lewis Research Center, Cleveland, Ohio 44135) and James Gibb (Defence Research Agency, Bedford MK416AE, England, United Kingdom). *Journal of Aircraft* Vol. 35, No. 4, July-August 1998.

[5] Experimental Investigation of Actuators for Flow Control in Short Inlet Ducts Authors: John C. Vaccaro, Yossef Elimelech and Michael Amitay. Rensselaer Polytechnic Institute, Troy, NewYork, 12180, USA. https://doi.org/10.2514/6.2010-862

[6] Active Flow Control Of A Diffusing S-Duct Author: Yahya Hamza Shanan *International Journal of Engineering, Science and Technology* Vol.12, No.2, 2020, pp.110 http://dx.doi.org/10.4314/ijest.v12i2.1 design and Numerical Study of Serpentine Duct Using Flow Controllers for Flow Characteristics Enhancement School of Mechanical Engineering, Reva University, Page 54

[7] Computational Design Optimization for S-Ducts Authors: Alessio D'Ambros, Timoleon Kipouros, Pavlos Zachos, Mark Savill and Ernesto Beninihttps://doi.org/10.3390/designs2040036

[8] Comparative Studies On Flow Control In Rectangular S-Duct Diffuser Using Submerged-Vortex Generators Authors: Akshoy Ranjan Paul, Pritanshu Ranjan, Vivek Kumar Patel, Anuj Jain Aerospace Science and Technology http://dx.doi.org/10.1016/j.ast.2012.11.014

[9] Design Of Vortex Generator Flow Control In Inlets Author: Adam Jirasek Swedish Defence Research Agency, FOI, 164 90 Stockholm, Sweden. *Journal of Aircraft* Vol.43, No.6,November-December2006. https://doi.org/10.2514/1.21364

[10] Navier-Stokes Analysis Of Three-Dimensional S-Ducts Authors: GJ Harloff, CF Smith, E Bruns and R DeBonis Sverdrup Technology, Inc., Brook Park,Ohio 44142 *Journal of Aircraft* Vol.30, No.4, July- August 1993. https://doi.org/10.2514/3.46375

[11] Passive Flow Control Study In An S-Duct Using Stereo Particle Image Velocimetry Authors: Geoffrey Tanguy, David G MacManus, PavlosZachos, Daniel Gil-Prieto and Eric Garnier Cranfield University, Cranfield, England MK43 0AL, United Kingdom and ONERA-The French Aerospace Lab, 90 Meudon, France. https://doi.org/10.2514/1.J055354