International journal of Engineering Research-Online
A Peer Reviewed International Journal
Articles available online http://www.ijoer.in

Vol.2., Issue.4, 2014

RESEARCH ARTICLE

ISSN: 2321-7758

DESIGN OF S-SHAPED INLET FOR AIRCRAFT ENGINES AND ANALYSIS BY USING FLUENT

SATHEISH KUMAR.M[1],AYYAPPAN.S[2]

[1]PG-Scholar,[2]Assistant Professor
Department of Mechanical Engineering
Government College of Engineering
Salem, Tamilnadu, India.

Article Received: 07/08/2014 | Article Revised on: 22/08/2014 | Article Accepted on:24/08/2014

SATHEISH KUMAR.M

ABSTRACT

The current trend in the design of missiles or combat aircraft is to reduce costs and to enhance the stealth. Hence the intake integration becomes an increasingly important issue. A S-shaped diffusing duct is an essential feature of a combat aircraft intake system. The basic shape of the duct is important since an engine requires air at a moderate subsonic speed. The primary purpose of the S-duct is to convey air from the wing or fuselage intake to the engine compressor. Further, it decelerates flow velocity and subsequently increases pressure head the kinetic energy head along its length. So we have designed two different shapes of S-Shaped inlets namely, S shaped inlet with longer duct length, Large Area ratio the short duct length and reduced bending. We have analyzed these two designs and final reduction in velocity an increase in pressure is achieved.

Keywords- Geometry of S-Shaped Duct,CFD,Analysis of various shapes.

.

INTRODUCTION

The engine inlet duct is to provide the engine compressor with a uniform supply of air in order to prevent the compressor from stalling. Since the inlet is directly exposed to the impacting airflow, it must also create as little drag as possible.

The S-duct is to convey air from the wing or fuselage intake to the engine compressor. Further, it decelerates flow velocity and subsequently increases pressure head the kinetic energy head along its length. The diffusion phenomenon is the conversion of kinetic energy of the fluid into pressure energy in the direction of flow.

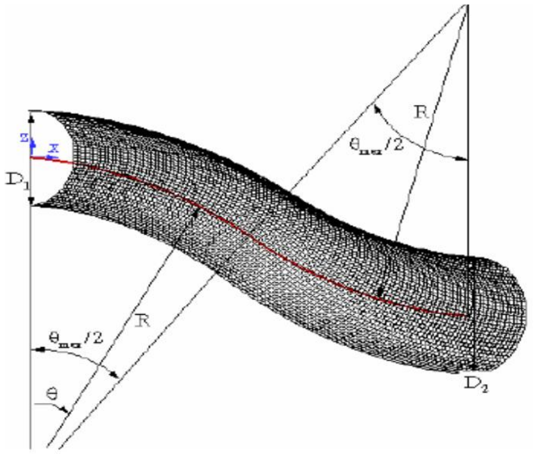
The developments in technology and reduction in radar cross section requirements are leading to serpentine inlet ducts that are shorter and have larger height offsets throughout the duct.

The advantages of S-Shaped inlet designs are to improve aircraft thrust to weight ratio, the serpentine shape of the inlet duct is becoming more aggressive as an ultra-compact, highly offset inlet duct, allowing a reduction in the overall aircraft length. While these inlet ducts reduce the radar cross section, they also reduce the stability margin of the propulsion system. The more aggressive inlet ducts result in increased turning which in turn produces secondary flow within the duct. The compact nature of the duct limits the length for diffusion and dissipation of these secondary flows and leads to greater distortion levels at the engine fan face.

This inlet distortion can produce a reduction in stability margin for the compressor or fan of the turbine engine. In addition, inlet distortion can result in high cycle fatigue, which may lead to catastrophic loss of aircraft, loss of aircraft operability, and increased maintenance costs. An S – shaped inlet can essentially block the line of sight of a compressor intake, thus sealing it from incoming radar waves along with the prevention of the heat signature being sent out the hot gases produced in the combustor.

I. GEOMETRY OF S-SHAPED DUCT

A well-designed diffusing duct should efficiently decelerate the incoming flow, over a wide range of incoming conditions, without the occurrence of stream wise flow separation. A short duct is desired because of space constraint and aircraft weight consideration, resulting in high degrees of centerline curvature.



The centerline curvature gives rise to streamline curvature causing cross-stream pressure gradients. These cross-stream pressure gradients impart a transverse or cross flow velocity, known as secondary flow to the fluid within the boundary layer. The axial development of the secondary flow in the form of counter rotating vortices at the duct exit is responsible for a good deal of flow non-uniformity at the engine face.

II. METHODOLOGY

CFD Analysis Process

In order to conduct a CFD analysis, three main tasks must be completed: grid generation, or pre-processing, the actual computational processing of the analysis, and visualization of the computational results, or post-processing. As CFD matured over the past few decades, each of these steps has become a discipline in and of itself.

To create 3d model in a CAD software and we have selected CATIA V5 for modeling. Since CATIA is familiar for aerospace designs we have selected this for our project. The shape and dimensions of the model we have used in our project is a research paper by NASA Lewis research center, Cleveland, Ohio.

We have designed the s-shape inlet with the help of multi section solid option in the CATIA part modeling. And we saved the file in the igs format for importing to other software's for meshing and finally to analyses the pressure variation by using fluent.

III. ANALYSIS PROCEDURE

Analysis of 22.5º/22.5º S-Shaped Diffuser:

Curved diffusers, which are widely used as aircraft intake ducts, accomplish certain objectives for prescribed inlet flow conditions within geometric limitations. On the basis of the available literature, it is seen that investigations on circular cross-section S-diffusers with area ratio greater than 1.5 have not been reported. The effect of turn angle (Δβ) in such diffusers is still to be investigated. An attempt has been made in the present study to establish the performance of an S-diffuser with fixed centerline length of 264 mm and inlet velocity of 40 m/s, using the commercial CFD code FLUENT. The predictions have been carried out for steady and incompressible flow. On a circular cross-section 22.5°/22.5° S-diffuser with *AR* = 1.5.

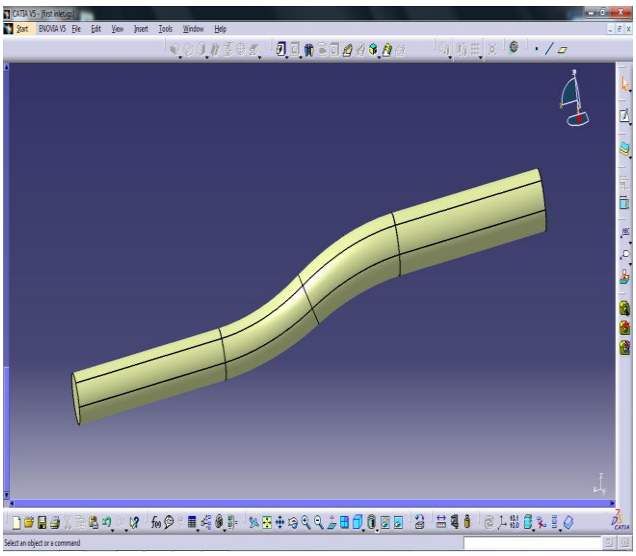
Parameter	*Value/Type*
Cross-section	*Circular*
Inlet diameter	*48.0 mm*
Outlet diameter	*58.8 mm*
Radius of curvature	*336.0 mm*
Centerline length of diffuser portion	*264.0 mm*
Upstream straight inlet portion	*210.0 mm*
Down-stream straight portion	*210.0 mm'*
Inlet velocity	*40.0 m/sec*

IV. S-SHAPED MODEL DESIGN USING CATIA

A. Designed model - 1

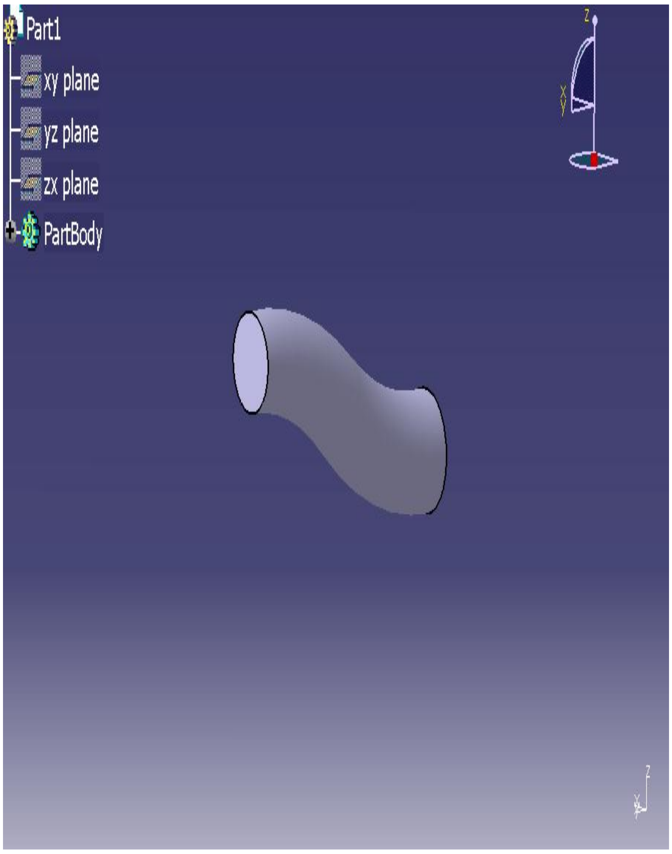
The flow analysis is done at operating conditions of 10,000 fts for the mach number of 0.7. The duct was designed in CATIA. The duct specifications are given below.

The duct centerline is defined by two circular arcs with an identical radius of curvature, *R* = 336.0 cms and subtended angle 30 **°.** *Dl* = 48.0 cms is the radius at the duct inlet and *D2* = 58.8 cms is the radius at the duct exit. This provides an exit the **inlet** area **ratio** of A_2/A_1 = 1.3.



B. Designed model - 2

The second analysis, duct a larger area ratio with short duct length with reduced bend. The flow analysis is done at operating conditions of 10,000 fts for the mach number of 0.7. The duct was designed in CATIA. The duct specifications are identical radius of curvature, *R* = 102.1 cms and subtended angle 30 °.
DI = 20.4 cms is the radius at the duct inlet and *D2* = 25.1 cms is the radius at the duct exit. This provides an exit the **inlet** area **ratio** of A_2/A_1 = 1.52.

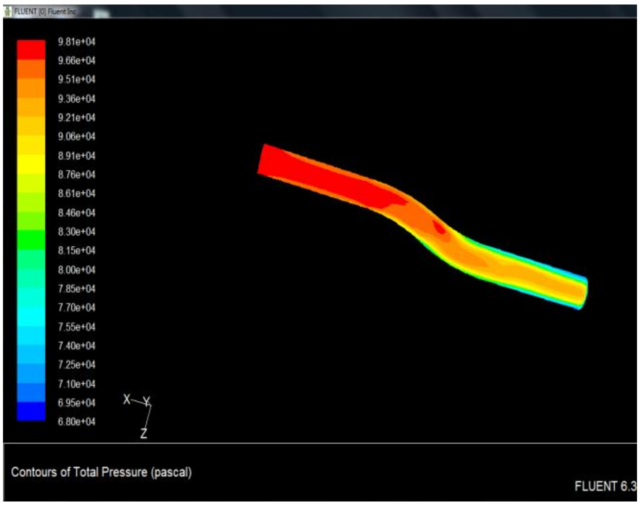


V. FLUENT ANALYSIS

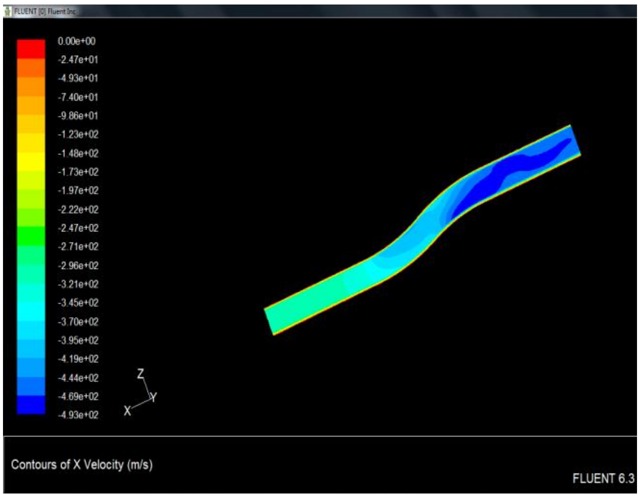
FLUENT is a state-of-the-art computer program for modeling fluid flow and heat transfer in complex geometries. It provides complete mesh flexibility, solving flow problems with unstructured meshes that can be generated about complex geometries with relative case. Supported mesh types include 2D triangular / quadrilateral, 3D tetra-hedral / hexa hedral / wedge and mixed meshes. FLUENT also allows refining or coarsening the grid based on the flow solution. A Fluent is based on Finite volume method. The finite volume method is a numerical method for solving partial differential equations that calculates the values of the conserved variables averaged across the volume. Advantage the finite volume method is, it does not require a structured mesh as in the case the Finite element method. It is preferable comparing with other methods as a result of the fact that boundary conditions can be applied non-invasively. This is true because the values of the conserved variable are located within the volume elements, and not at nodes or surfaces. Finite volume methods are specially powerful on coarse non-uniform grids and in calculations where the mesh moves to track interfaces or shocks.

ANALYSING OF MODEL

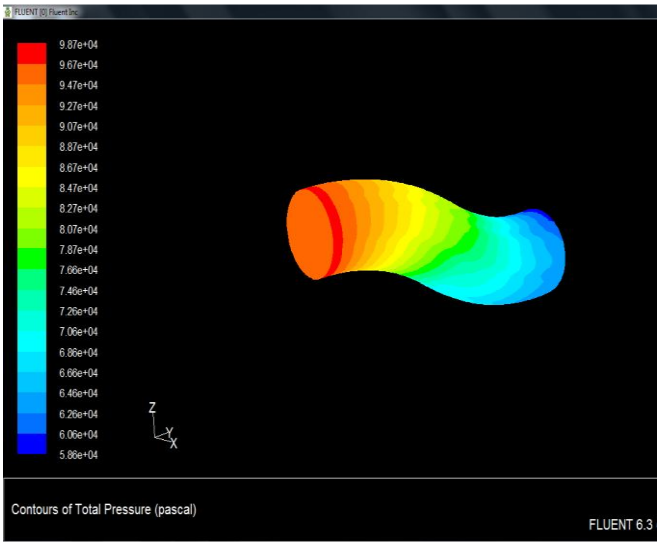
A. Variation of total pressure in model - 1



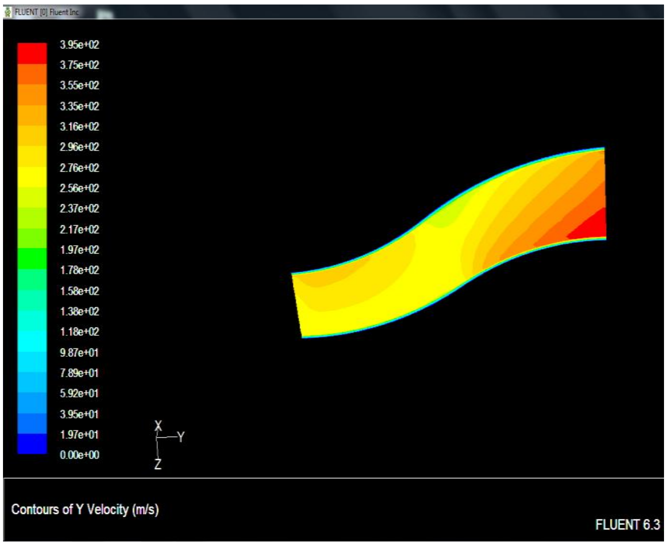
B. Variation of X velocity in model - 1



C. Variations of total pressure in model - 2



D. Variation of Y velocity in model – 2



VII. RESULTS

(I) VARIATION OF TOTAL PRESSURE

There was a considerable recovery the total pressure at the exit. The use of short duct length inlet improve the flow let to flow deceleration at the leading edge of guide vane.

Pressure at inlet =98,700 pa

Pressure at outlet =58,600 pa

(ii) **Variation of static pressure**

Static pressure variation along the upper wall was higher than the lower wall variation.

Pressure at inlet =6530 pa

Pressure at outlet =2170 pa

(iii) **Variation of velocity along duct axis**

Velocity along duct axis increases due to turbulence.

Velocity at inlet =276 m/s

Velocity at outlet =315 m/s

This increase in velocity leads to shock wave formation which reduces pressure at outlet

CONCLUSION

The various configuration of s-shaped duct were studied. Various modification were applied for flow improvement. From the numerical studies carried out on circular cross-section S-diffusers the following major conclusions are drawn:

1. Counter-rotating vortices are present throughout the diffuser length with magnitudes being higher for the higher angle of turn.
2. Pressure recovery is observed to be a weak function of angle of turn for the diffusers investigated.
3. Increase in the area ratio of the 22.5°/22.5° diffuser increases the static pressure recovery from 44.72% to 74.04%, for increase in the area ratio from 1.5 to 3.0.
4. The counter-rotating vortices obtained for different area ratio diffusers are of unequal size. The magnitude of the cross-velocity decreases with the increase in the area ratio. For 3.0 area ratio diffuser, there is tendency of vortices to break up into larger numbers. The factor which affects the flow in s-shaped duct are

- Duct length
- Bend radius

- Area ratio
- Vorticity at bends

By conducting studies in these areas better design configuration can be obtained for the improved flow.

REFERENCES

[1]. M. Rowe, "Measurements and Computations of Flow in Pipe Bends", J. Fluid Mechanics, **43** (1970), pp. 771–783.

[2]. P. Bansod and P. Bradshaw, "The Flow in S-shaped Ducts", Aeronautical Quarterly, **23** (1972), pp. 141–145.

[3]. L.A. Butz, "Turbulent Flow in S-shaped Ducts", M.Sc. Thesis, Purdue University, USA, 1979.

[4]. A.M.P.K. Taylor, J.H. Whitelaw, and M. Yianneskis, "Developing Flows in S-shaped Ducts. Part 1: Square Cross-section", Report No. FS/81/22, Mechanical Engineering Department, Imperial College of Science, Technology and Medicine, London,U.K., 1981.

[5]. A.M.P.K. Taylor, J.H. Whitelaw, and M. Yianneskis, "Developing Flows in S-shaped Ducts. Part 2: Circular Cross-section", NASA Contract Report 3759, 1984.

[6]. J. Rojas, J.H. Whitelaw, and M. Yianneskis, "Flow in Sigmoid Diffusers of Moderate Curvature", Report No. FS/83/21, Mechanical Engineering Department, Imperial College of Science, Technology and Medicine, London, U.K., 1983.

[7]. R.W. Guo and J. Seddon, "The Swirl S-duct of Typical Air Intake Proportions", Aeronautical Quarterly, **32** (1983), pp. 99–129.

[8]. R.W. Guo and J. Seddon, "Swirl Characteristics of an S-shaped Air Intake with Both Horizontal and Vertical Offsets", Aeronautical Quarterly, **32** (1983), pp. 130–146.

[9]. J. Seddon "Understanding and Countering the Swirl in S-ducts: Tests on the Sensitivity of Swirl Fences", Aeronautical Journal, **88** (1984), pp. 117–127.

[10]. O.J. McMillan, "Mean-flow Measurements of the Flow Field Diffusing Bend", NASA Contract Report 3634, 1982.