The International Journal Of Engineering And Science (IJES)
||Volume|| 3 ||Issue|| 01 || Pages || 53-59 || 2014 ||
ISSN (e): 2319 – 1813 ISSN (p): 2319 – 1805

CFD Study of Air Intake Diffuser

Prasath M S[1], Shiva Shankare Gowda A S[2], Senthilkumar S[3]
[1] Post Graduate Scholar, Department of Aeronautical Engineering,
[2] Professor, Department of Aeronautical Engineering,
[3] Assistant Professor, Department of Aeronautical Engineering,
Nehru Institute of Engineering and Technology, T.M. Palayam, Coimbatore – 641 105, Tamil Nadu, India.

ABSTRACT
Aircraft propulsion systems often use subsonic diffusing ducts as air-intakes to supply the atmospheric air to the engine compressor. Due to space constraint, the diffusers need to be curved; this causes severe flow non-uniformity at the engine face. The air intake of the aircraft supplies the mass flow demand of the engine over a range of aircraft speeds and altitudes with high pressure recovery and minimum total pressure loss at the engine face. Also the duct must deliver air to the compressor under all flight conditions with a little turbulence. This paper attempts to study the flow inside diffusing duct and the pressure distribution at the AIP. The air intake duct was designed using CATIA. The meshing and analysis of the duct was accomplished using ICEM-CFD and CFX respectively.

Keywords *- air intake, duct, diffuser, AIP.*

Date of Submission: 24 December 2013 — Date of Acceptance: 10 January 2014

LIST OF SYMBOLS

AIP	Aerodynamic interface plane
C_{TL}	Total pressure loss coefficient
C_{PR}	Static pressure recovery coefficient
p_{ti}	Total pressure at inlet
p_t	Total pressure at any point
p_s	Static pressure at inlet
p_{si}	static pressure at inlet
ρ	Air density
U_{avi}	Average inlet velocity
Re	Reynolds number
D_H	The hydraulic diameter of the pipe
v	The mean velocity of the object relative to the fluid
μ	Dynamic viscosity of fluid
β	Turning angle

I. INTRODUCTION

The air intake is that part of an aircraft structure by means of which the aircraft engine is supplied with air taken from the outside atmosphere. The air flow enters the intake and is required to reach the engine face with optimum levels of total pressure and flow uniformity. These properties are vital to the performance and stability of engine operation. Depending on the type of installation, this stream of air may pass over the aircraft body before entering the intake properly. Selection of the correct type of intake and the associated intake geometry has important consequences to any airplane design. For that reason, intake design receives considerable attention in the design phase of an airplane. Intakes come in a variety of shapes and sizes with the specifics usually dictated by the speed of the aircraft. An engine's air intake duct is normally considered an airframe part and made by aircraft manufacturer. During flight operation, it is very important to the engine performance. Engine thrust can be high only if the intake duct supplies the engine with the required airflow at the highest possible pressure.

Intakes must be able recover as much of the total pressure of the free air stream as possible and deliver this pressure to the front of the engine compressor. The duct must deliver air to the compressor under all flight conditions with a little turbulence. As far as the aircraft is concerned, the duct must hold to a minimum of the drag. The duct also usually has a diffusion section just ahead of the compressor to change the ram air velocity into higher static pressure at the face of the engine. This is called ram recovery. The intake duct is built generally in the divergent shape (subsonic diffuser).

II. LITERATURE SURVEY

K. Saha, S. N. Singh, and V. Seshadri and S. Mukhopadhyay, "Computational Analysis on Flow through Transition S-Diffusers: Effect of Inlet Shape", JOURNAL OF AIRCRAFT, Vol. 44, No. 1, January–February 2007- Studied various cross section shapes of the inlet, namely, elliptic, semi-circular, oval, rectangular, and square, have been analysed using the standard k–ϵ turbulence model. For all the ducts, the angle of turn (22:5=22:5 deg), the centerline length (300 mm) and the circular exit diameter (100 mm) have been kept constant. Incompressible flow analysis has been done at 0.17 Mach number at the entry of the duct which corresponds to the throat of intakes. The elliptic-shaped inlet duct shows the best performance in terms of pressure recovery, loss coefficient, and flow distortion at the compressor face, whereas square duct produces the worst flow characteristics. It is also established that the renormalized group k–ϵ model predicts better than the standard k–ϵ turbulence model. The functions of a well-designed diffusing duct are to decelerate the flow efficiently and increase the static pressure with minimal total pressure loss and distortion at compressor face of the engine. The S-shaped intake ducts invariably have circular outlet whereas inlet shape has to be made compatible with the fuselage shape and location. The inlet shape optimization for compatibility with the engine is one of the major issues in the S-duct design.

P. E. H. Abrahamsent, B. A. Pettersson Reifi, L. Szetrar, G J. B. Fossdalt, "Air Intake Studies: Experimental measurements and computational modelling"-Studied the flow in an S-shaped air intake using experimental and computational methods. In the experimental studies the measurements of an isentropic light piston tunnel has been carried out, whereas in computational studies the non-linear eddy viscosity model has been compared with a linear counterpart. The combination of a highly turning S-duct and a significant divergence of the cross-sectional area makes the flow susceptible to separation which leads to low total pressure recovery and thereby reduced engine performance. The results have been compared with available experimental data. CFD analysis was done. – K-ϵ turbulence model was used for the study. The flow field exhibits a separation along the starboard side. A pair of vortices can also be observed close to the top and bottom side of the air intake. It is found that both computations and experiments predict the pressure recovery in a satisfactory way.

III. DESCRIPTION OF METHODOLOGY

3.1 . Computational modeling

CFD codes are traditionally based on the Reynolds Averaged Navier Stokes equations which require a model for the unknown single point correlation of fluctuating velocities, i.e. the so called Reynolds stresses. A viable approach to air intake design is to apply less sophisticated turbulence models to the external flow calculations which then provide inlet conditions for internal flow simulations, thus taking into account effects of fuselage boundary layers, etc. The internal flow computations, which inevitably require more complex turbulence closures, can then be applied to a smaller computational domain so that the computational cost is significantly reduced. The flow inside the air intake is mainly characterized by three-dimensional turbulent boundary layers, affected by an adverse pressure gradient, and a potential core flow. In addition laminar-turbulent transition and a mean swirling motion is also most likely present. It is therefore unfortunate that the majority of commonly used CFD tools today adopt turbulence closures based on Boussinesq's linear stress-strain relationship. An inherent shortcoming of these closures (e.g. the standard k - E model) is that they are unable to capture direct effects of body forces on the turbulence.

3.2. Turbulence Models

In studying turbulent flows, the objective is to obtain a theory or a model that can yield quantities of interest, such as velocities. For turbulent flow, the range of length scales and complexity of phenomena make most approaches impossible. The primary approach in this case is to create numerical models to calculate the properties of interest. A selection of some commonly-used computational models for turbulent flows is presented in this section.

The chief difficulty in modeling turbulent flows comes from the wide range of length and time scales associated with turbulent flow. As a result, turbulence models can be classified based on the range of these length and time scales that are modeled and the range of length and time scales that are resolved. The more turbulent scales that are resolved, the finer the resolution of the simulation, and therefore the higher the computational cost. If a majority or all of the turbulent scales are modeled, the computational cost is very low, but the tradeoff comes in the form of decreased accuracy.

In addition to the wide range of length and time scales and the associated computational cost, the governing equations of fluid dynamics contain a non-linear convection term and a non-linear and non-local pressure gradient term. These nonlinear equations must be solved numerically with the appropriate boundary and initial conditions.

IV. DESIGN AND ANALYSIS

4.1. Design of the duct

Due to space constraint the diffusers need to be curved. If the limbs of the air intake tend to be circular in cross section then it becomes difficult for the two limbs to merge at a plane. Hence a rectangular cross section was chosen for the air intake and exit.

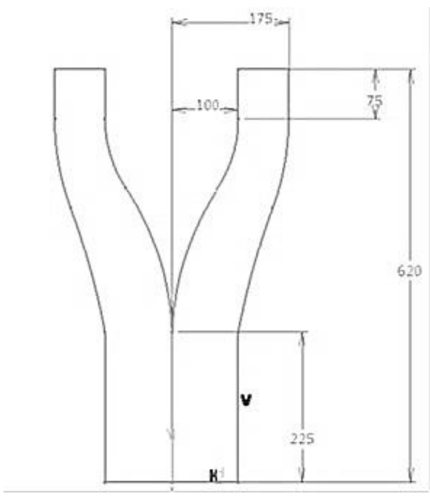

Fig.1.Y-Duct intake

Thus the Y duct diffuser is designed as per Fox and Kline and is based on linear area ratio from inlet to exit. The inlet area of the test diffuser was chosen as 75x75 mm^2 .Straight length of 75 mm is added to both the inlets and the outlet for proper boundary layer growth. Both the inlets have a turning angle of 20^o and the area of both the limbs increases till the two limbs at a plane. From this plane the duct is tapered to an outlet width of 200 mm. The area ratio of the Y duct is 1.33 which is calculated from inlet to outlet of the duct. Fig.1 shows the Y duct as per Fox and Kline.

4.2. Software Used

CATIA V5R17 was used to design the Y duct. Fig.2 shows the three dimensional view of the Y duct.

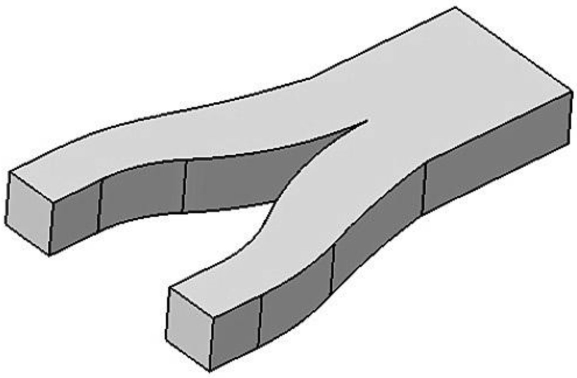

Fig.2.Three dimensional view of the Y-Duct

4.3. Mesh Generation

For meshing the Y duct ICEM CFD was used. The geometry was divided into different parts to define different boundary regions. The boundary regions for the duct are the inlet, outlet, wall, inner wall and outer wall. The maximum mesh size used was 5. For the walls we have used prism mesh and for other parts tetra mesh was used.

Table.1.Mesh number

Domain	Nodes	Elements
Fluid	90291	465655

After the mesh parameters were set, the solver output file was created. The output solver and the output structural solver were selected as ANSYS CFX and ANSYS.

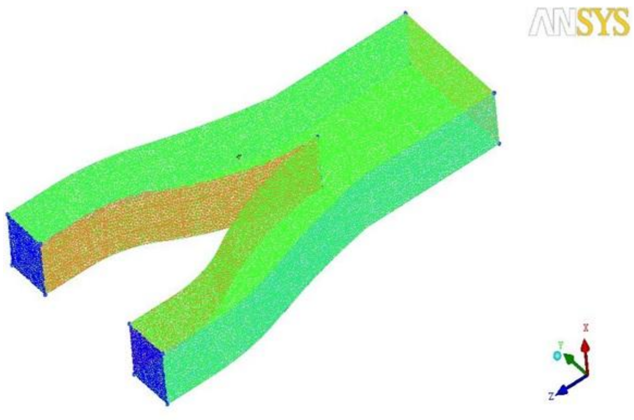

Fig.3.Mesh generated over a duct

V. ANALYSIS

The analysis was carried out using CFX. In the Pre- processing the following boundary conditions were defined. First we tested the bare duct for zero angle of attack. The inlet velocity is 19.67m/s. in the second module the angle of attack will be increased and tested. The simulation is started and the equations are solved iteratively as a steady-state or transient. Finally a postprocessor is used for the analysis and visualization of the resulting solution.

5.1. Boundary Conditions

Table.2.Boundary conditions

Inlet	
Type	INLET
Location	INLET
Flow Regime	Subsonic
Heat Transfer	Static Temperature
Static Temperature	3.0000e+02 [K]
Mass And Momentum	Normal Speed
Normal Speed	1.9670e+01 [m s^-1]
Turbulence	Medium Intensity and Eddy Viscosity Ratio
Outlet	
Type	OUTLET
Location	OUTLET
Flow Regime	Subsonic
Mass And Momentum	Average Static Pressure
Pressure Profile Blend	5.0000e-02
Relative Pressure	0.0000e+00 [Pa]
Pressure Averaging	Average Over Whole Outlet
Wall	
Type	WALL
Location	SIDEWALL, WALL, INNERWALL
Heat Transfer	Adiabatic
Mass And Momentum	No Slip Wall
Wall Roughness	Smooth Wall

5.2. Domain

Table.3.Domain values

Fluid	
Type	Fluid
Location	FLUID
Materials	
Air Ideal Gas	
Morphology	Continuous Fluid
Settings	
Buoyancy Model	Non Buoyant
Domain Motion	Stationary
Reference Pressure	1.0000e+00 [atm]
Heat Transfer Model	Total Energy
Turbulence Model	k epsilon

5.3. Observation

From Fig.4 we can see flow separation occurring at the inflexion plane and turbulence at the diffuser exit. The dark region in the Figure-4 shows the flow separation. Fig.5 shows non uniform flow at AIP. If this flow is let into the compressor they damage the blades.

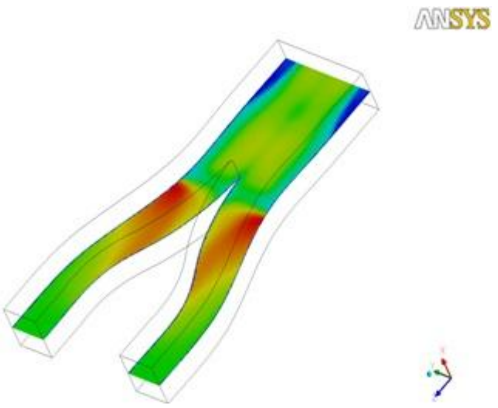

Fig.4. Bare duct showing flow separation at the inflexion plane

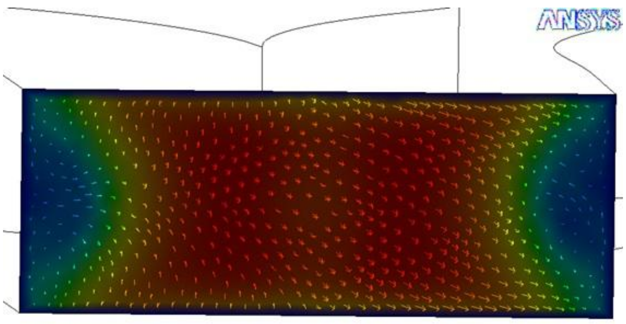

Fig.5. Bare duct showing non uniform flow at AIP

VI. RESULTS AND SOLUTION

These vortices cause a decrease in the intake efficiency. The divergence of the duct allows the separation point to shift further downstream. Shifting the separation point downstream enables the expanded airflow to persist proportionality longer, the flow velocity at the separation point to become slower and consequently the static pressure to become higher. The static pressure at the separation point governs over all the pressures in the entire flow separation region. It shifts the separation point downstream therefore raises the pressure of the flow separation region.

REFERENCE

[1] Abrahamsent.P.E.H, Pettersson Reifi. B.A, Szetrar. L, Fossdalt. J.B, "Air Intake Studies: Experimental measurements and computational modelling".
[2] Akshoy Ranjan Paul, Pritanshu Ranjan, Vivek Kumar Patel, Anuj Jain, "Comparative Studies On Flow Control In Rectangular S-Duct Diffuser Using Submerged-Vortex Generators"- Aerospace Science and Technology 28 (2013) 332–343.
[3] R.W. Fox, and S.J. Kline, Flow Regimes in Curves Subsonic Diffuser. Trans ASME, Journal of Basic Engineering, Vol. 84, 1962, pp. 303 – 316
[4] Frederic Smith.C, Steve D. Podleskif, Wendy S. Barankiewicz and Susan Zeleznik.Z, "Comparison of F/A-18A Inlet Flow Analyses with Flight Data Part 2"- Journal of Aircraft, Vol. 33, No. 3, May-June 1996.
[5] Saha.K, Singh.S.N, Seshadri.V, and Mukhopadhyay.S, "Computational Analysis on Flow through Transition S-Diffusers: Effect of Inlet Shape"- Journal of Aircraft, Vol. 44, No. 1, January–February 2007.
[6] Stanley R. Mohler Jr, "Wind-Us Flow Calculations For The M2129 S-Duct Using Structured And Unstructured Grids"- (AIAA-2004-0525)
[7] Mattingly, J.D. (2006) 'Elements of Propulsion : Gas Turbines and Rockets'.
[8] Oates, G.C. (1985) 'Aerothermodynamics of Aircraft Engine Components'.
[9] Saravanamuttoo, Cohen, H., Rogers, GFC., HIH. (1996) 'Gas Turbine Theory'.