Computational Fluid Dynamics Simulation and Optimization of a Gas Turbine Inlet Duct

Mahendra M. Dhongadi [1]*, K.V. Sreenivas Rao,[2] G.R. Srinivas,[1] C.P.S. Prakash[1]

[1] *Department of Mechanical Engineering, Dayananda Sagar College of Engineering, Bangalore.*
[2] *Department of Mechanical Engineering, Siddaganga Institute of Technology, Tumkur.*

Abstract

The overall aim of this paper is to identify the best inlet for gas turbine, which will have the minimum pressure loss and well guided flow profile. The experimental and computational studies are carried out on sharp edge inlet, convergent inlet at various semi angles and bell mouth inlet at various bend radiuses. The experimental method is carried out to find the velocities of various Inlet Duct models by using hot wire anemometer in order to identify its Total Pressure Losses (ΔP). In this project work, 3D Finite Volume Models and Computational Fluid Dynamics (CFD) analyses of inlet ducts are used to determine the Total Pressure loss (ΔP). A progressively refined tetrahedral finite element mesh is created using Pre-processing procedure in ANSYS software. ANSYS CFX-Pre-processor is employed to define the properties of flow and simulation can be obtained from ANSYS CFX-Postprocessor. From the Experimental and CFD results, we can conclude that bell mouth inlet produces a minimum pressure loss and uniform flow compared to Sharp edge and Converging Inlets. The present work on inlet ducts is useful in engine test cells, intake runner of reciprocating engine and on engines installed in helicopters.

Keywords**:** *Computational Fluid Dynamics, Gas Turbine, Axial Ducted Fan, Turbomachinery.*

1. Introduction

The gas turbine is an internal combustion engine that uses air as the working fluid. The engine extracts chemical energy from fuel and converts it to mechanical energy using the gaseous energy of the working fluid (air) to drive the engine and propeller, which in turn propel the airplane [1],[2]. As air passes through a gas turbine engine, aerodynamic and energy requirements demand changes in the air's velocity and pressure. During compression, a rise in the air pressure is required, but not an increase in its velocity. After compression and combustion have heated the air, an increase in the velocity of gases is necessary in order for the turbine rotors to develop power. The size and shape of the ducts through which the air flows affect these various changes. Where a conversion from velocity to pressure is required, the passages are divergent. Conversely, if a conversion from pressure to velocity is needed, a convergent duct is used [3].

2. Methodology

2.1 Initial thinking

It is very important to understand as much as possible about the problem being simulated in order to accurately define it. This stage involves collecting all the necessary data required for the simulation including geometry details, fluid properties, flow specifications, boundary conditions and initial conditions.

2.2 Pre-processor

Build geometry: Depending on whether the problem geometry is one, two or three dimensional, the geometry consists of creating lines, areas or volumes. The geometry of the flow domain is created using ANSYS Design Modeller.

Define materials: A material is defined by its material constants. Every volume has to be assigned a particular material.

Mesh generation: In this stage, the continuous space of the flow domain is divided into sufficiently small discrete cells. The distribution of which determines the positions, where the flow variables are to be calculated and stored. Variable gradients are generally more accurately calculated on a fine mesh than on a coarse one. A fine mesh is therefore particularly important in regions where large variations in the flow variables are expected.

Flow specification: The solver is a specialised programme that solves the numerical equations based on the data specified in the data file. The results obtained by the solver are written to a results file for examination using the post-processor software.

2.3 CFX-Solver

All the data defined in the pre-processing steps are fed into the solver 2-D sketches are first drawn and 3-D tools are then used to generate the full geometry.

2.4 CFD analysis in CFX-Postprocessor

In this software, the data obtained by the solver can be visualised and displayed.

3. Duct Design

The objective of duct design is to size the duct, so as to minimize the pressure drop through the duct, while keeping thesis of the duct to a minimum. Proper duct design requires knowledge of the factors that affects pressure drop and velocity in the duct. the principle of divergent duct, where energy is neither being added or taken away, but where the gaseous energy is being converted from velocity to pressure and temperature. There is a velocity decrease as air flows from a small inlet to a larger outlet. As velocity decreases, impact pressure (Pi) also decreases. Since no energy is added or subtracted from the system, total pressure (Pt) for the air remains constant and static pressure (Ps) increases. One way of viewing this is that the impact pressure is converted to static pressure; thus, a static pressure rise is seen as air flows through a divergent duct and is compressed. The convergent duct operates exactly in reverse of the divergent duct. the principle of convergent duct, where energy is neither being added or taken away [4],[5].

3.1 Ducted Axial Fan Test Rig

An Inlet duct Dimensions of a Ducted Axial Fan (DAF) is as shown in Fig. 1, was used for Analytical and CFD simulation in this study. The circular duct is made of fiber reinforced polyester with a smooth internal finish. The motor is positioned inside a 390 mm diameter ✕ 457 mm length of fan casing.

Fig.1. Ducted Axial fan test rig

3.2 Different types of Inlet Ducts

The Inlet Ducts can be divided in to three main categories: Sharp edge inlet duct (SID), Convergent Inlet Duct (CID) and Bell mouth Inlet Duct (BID).

3.2a Sharp edge Inlet Duct

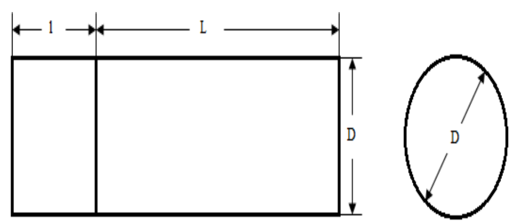

Fig. 2. Geometry of Sharp edge Inlet Duct

3.2b Convergent Inlet Duct

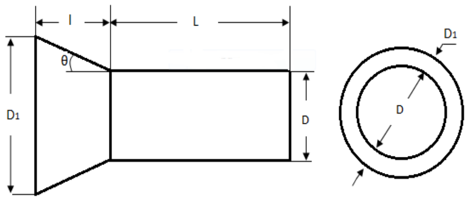

Fig. 3. Geometry of Convergent Inlet Duct

3.2c Bell mouth Inlet Duct

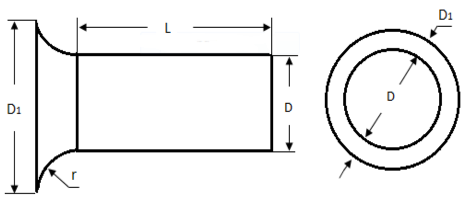

Fig. 4. Geometry of Bell mouth Inlet Duct

4. Experimental Method

Total Pressure Loss (ΔP) for various Duct Inlet models is calculated by the measurement of velocity through experimental method using hot wire anemometer [6], [7], [8]. There are two types of Losses in Ducts, major loss and minor loss. The sum of these losses is the Total Pressure Loss (ΔP).

Frictional Loss comes under major loss and Dynamic Losses are classified as minor loss. If the length of the duct is small the minor loss is greater than major loss. In the following pages Frictional and Dynamic Losses for various Inlet models namely, Sharp edge Inlet, Convergent and Bell mouth Inlet are find by experimental method.

4.1 Flow Loss study for various inlets

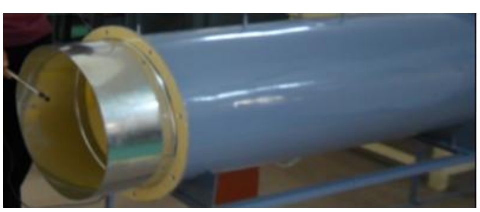

Fig. 5: Measurement of velocity by Hot Wire Anemometer

4.1a Sharp edge Inlet Duct

The velocity at Sharp edge Inlet duct measured experimentally using hot wire anemometer as shown in Fig.5. The measured velocity of Sharp edge inlet duct is 10m/s used to calculate its total pressure loss.

4.1b Frictional Losses

The pressure loss due to friction is known as frictional pressure drop or friction loss (ΔP_f). Pressure loss due to friction can be calculated by the Darcy equation given by the equation.

$$\Delta P_f = {4f\rho LV^2}/{2D}$$

$$\Delta P_f = \frac{0.0144 \times 1.2 \times 1.15 \times 10^2}{2 \times 0.39}$$

$$\Delta P_f = 2.54\ N/m^2$$

4.1c Dynamic Losses

Dynamic losses result from flow disturbances caused by duct mounted equipment and fittings that change the airflow path's direction and/or area. The Dynamic Losses in terms of total pressure.

$$\Delta P_i = C_o \times (\rho V^2/2)$$

$$\Delta P_i = 0.5 \times \left(\frac{1.2 \times 10^2}{2}\right)$$

$$\Delta P_i = 30\ N/m^2$$

The total pressure loss is taken as the sum of pressure loss due to friction loss and the dynamic loss.

$$\Delta P = \Delta P_f + \Delta P_i$$

$$\Delta P = 30 + 2.54$$

$$\Delta P = 32.54\ N/m^2$$

The Total Pressure Loss (ΔP) of a Sharp edge Inlet Duct (SID) is 32.54 N/m^2. Similarly, the Pressure Loss value calculated for Convergent Inlet Duct and Bell mouth Inlet Duct.

4.1d Experimental results of Convergent Inlet Duct

The Total Pressure Loss (ΔP) of a Convergent Inlet Duct (CID) at inlet angle (θ) 6° is 18.71 N/m^2. Similarly, the Pressure Loss is studied for various Convergent angles (8^o, 10^o and 12^o) experimentally and is listed in Table 1.

4.1e Experimental results of Bell mouth Inlet Duct

The Total Pressure Loss (ΔP) of a Bell mouth Inlet Duct (BID) at inlet radius (r) 97.5mm is 3.03

N/m^2. Similarly, the Pressure Loss is calculated for various Bell mouth radiuses (r) experimentally and is listed in Table 2.

Table 1: Experimental results of convergent duct

Sl. no	Convergent Inlet Angle (θ)	Friction Losses in terms of Total Pressure Loss of Convergent Inlet Duct
1.	6^o	3.31
2.	8^o	3.08
3.	10^o	2.99
4.	12^o	2.24

Table 2: Experimental results of Bellmouth duct

Sl. no	Bell mouth radius (r)	Friction Losses in terms of Total Pressure Loss of Bell mouth Inlet Duct $(\Delta P_f)+(\Delta P_{fb})$
1.	90	2.5
2.	95	2.38
3.	97.5	2.32
4.	105	2.38
5.	120	2.48

From the above set of experimental studies it is observed that bell mouth with 97.5mm bend radius has minimum pressure loss. Now this has to be checked with CFD results.

5. FV-MODELLING OF INLET DUCT

5.1 Modelling

Before any CFD processing can be done, a geometry model must be constructed. The use of ANSYS Design Modeller in this application was chosen for its ease in modelling. Generate a FV-Model using key-points, lines and circles. The FV-model is created based on a global coordinate system is Cartesian with x, y and z axis. The FV-Modelling of a Sharp edge Inlet Duct (SID) is generated to the dimensions.

5.2 Meshing

The most important part in CFD simulation is discretization of geometry. In order to calculate the fluid flow within the Duct to a reasonable accuracy, a good quality mesh needs to be constructed. Generally, hexahedral and tetrahedral meshes are used for CFD codes. ANSYS Work-Bench Mesh utilizes a tetrahedral mesh seen in fig. 7. for the mesh construction.

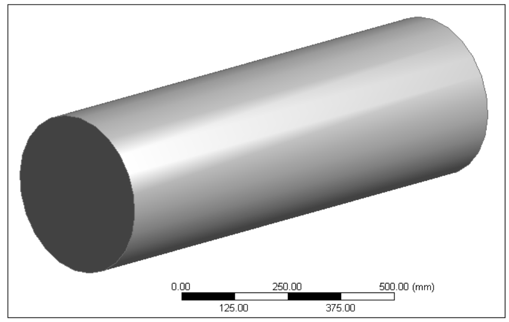

Fig 6: 3-D model of sharp edge duct

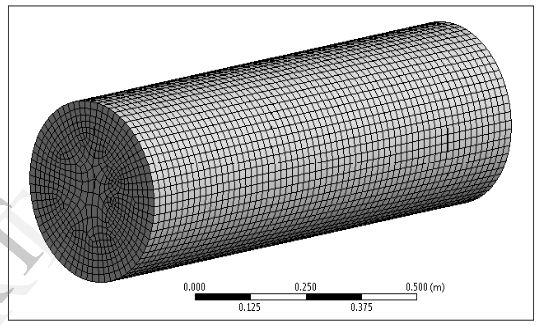

Fig 7: Mesh view of a sharp edge duct.

5.3 Boundary Conditions

The boundary conditions are applied at the selected area of the Sharp edge Inlet Duct (SID) as shown in Fig. 8.

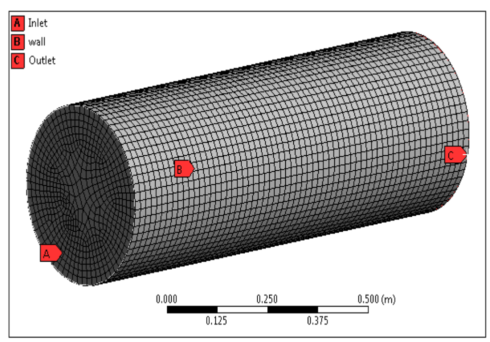

Fig 8: Applied boundary conditions on duct

6. Results and discussion

The Total Pressure Loss (ΔP) values of a Gas turbine Inlet Ducts are determined. Simulations are performed on FV-model using CFD (CFX-Post) analysis. The maximum Total Pressure Loss (ΔP) value occurs in Sharp edge Inlet Duct (SID) and minimum Total Pressure Loss (ΔP) occurs in Bell mouth Inlet Duct (BID).

6.1 Sharp edge Inlet Duct (SID)

The first case investigated is the Sharp edge Inlet Duct (SID) and using CFD (CFX-Post) a velocity contour and pressure contour plot were taken.

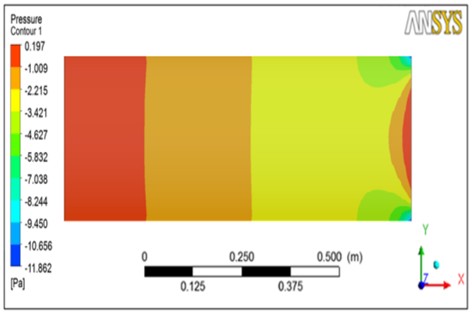

Fig 9: Pressure plot for Sharp edge Inlet Duct

The Total Pressure Loss (ΔP) value in a Sharp edge Inlet Duct (SID) is calculated by a functional calculator average area of Total Pressure, i.e. ΔP = 3.52Pa.

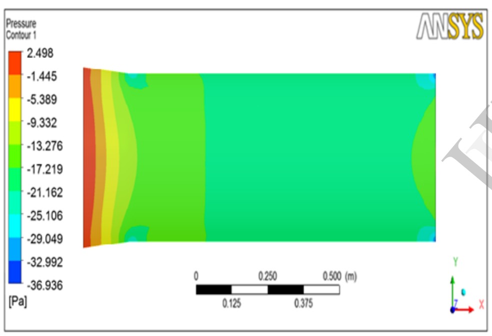

Fig10: Pressure plot for Convergent Inlet θ = 6°

6.2 Convergent Inlet Duct (CID) at inlet angle θ = 6°

The different cases of Convergent inlets were investigated with their own unique inlet angles **(θ).** The second case investigated was the Convergent Inlet Duct (CID) at inlet angle 6° and using CFD (CFX-Post) a velocity contour and pressure contour plot were taken.

The pressure contour plot in Fig. 10 shows pressure decreasing downstream of the Duct from the pressure drop located on the walls of the Duct. The Total Pressure Loss ΔP value in Convergent Inlet Duct (CID) at inlet angle 8° is calculated by a functional calculator average area of Total Pressure, i.e. ΔP = 2.94Pa.

6.3 Bell mouth Inlet Duct (BID) at inlet radius (r) 97.5 mm

The different cases of Bell mouth inlets were investigated each with their own unique inlet radius (r). The sixth case investigated was the Bell mouth Inlet Duct (BID) at inlet radius (r) 97.5 mm and pressure contour plot were taken is as shown in Fig. 11. The Total Pressure Loss ΔP value in Bell mouth Inlet Duct (BID) at inlet radius (r) 97.5mm is calculated by a functional calculator average area of Total Pressure, i.e. ΔP = 2.31Pa.

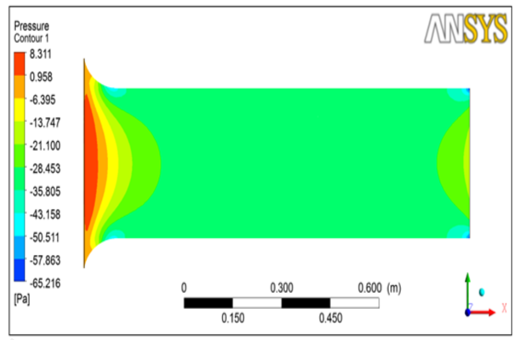

Fig 11: Pressure plot for Bellmouth Inlet r= 97.5mm

6.4 CFD and Experimental results comparison

The values computed from CFD simulation (CFX-Post) of Sharp edge Inlet Duct (SID) are compared with experimental results, as shown in Table 3. The experimental results and those obtained with a CFD simulation have 85% agreement with each other. The mean rate of error between the experimental and CFD result are 15%. The values computed from CFD simulation (CFX-Post) of a Convergent Inlet Duct (CID) are compared with experimental results as shown in Table 4 for varying converging angle (θ).

Tab 3: CFD and Experimental results comparison (SID).

CFD results (ΔP)	3.52
Experimental results (ΔP)	2.54

The experimental results and those obtained with a CFD simulation have 90% agreement with each other. The mean rates of errors between the experimental and the CFD results in Fig. 12 are 6% to 5% respectively. These errors can result from such things as selected turbulence and mathematical model, disregarding environment conditions and grid size in numerical models.

Tab 4: CFD and Experimental Results comparison (CID)

Convergent angle	6°	8°	10°	12°
CFD results	3.11	2.94	2.89	2.36
Experimental results	3.31	3.08	2.99	2.24

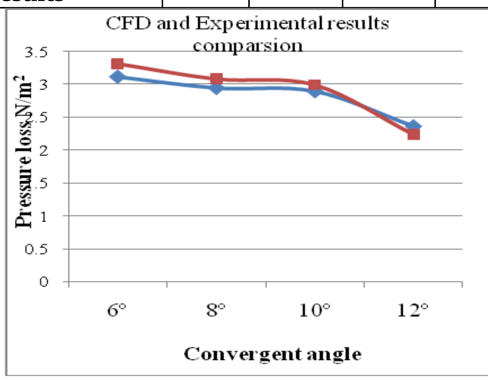

Fig. 12: Plot of ΔP v/s angle of Convergent Duct.

The values computed from CFD simulation (CFX-Post) of a Bell mouth Inlet Duct (BID) are compared with experimental results as shown in Table 5, for varying bell mouth radius (r). The experimental results and those obtained with a CFD simulation have 90% agreement with each other. The mean rates of errors between the experimental and the CFD results in Fig. 13 are 8% to 6% respectively. These errors can result from such things as selected turbulence and mathematical model, disregarding environment conditions and grid size in numerical models.

Tab 5: CFD and Experimental results comparison (BID).

Bell mouth radius	90	95	97.5	105	150
CFD results	2.4	2.3	2.3	2.4	2.4
Experimental results	2.5	2.3	2.3	2.3	2.4

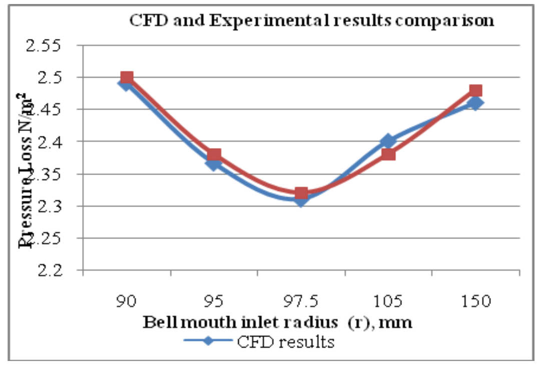

Fig. 13: Plot of ΔP v/s Bell mouth Inlet radius (r) mm of the duct Duct.

7. Conclusion:

The three-dimensional CFD simulation used to obtain Total Pressure loss ΔP of a Sharp edge Inlet Duct (SID), Convergent Inlet Duct (CID) and a Bell mouth Inlet Duct (BID). The Total Pressure Loss ΔP values of CFD results were compared with experimental solution. The experimental and CFD results are compared and optimized to identify the inlet having minimum pressure loss and uniform flow. From the Experimental and CFD results, we can conclude that bell mouth inlet is having a minimum pressure loss and uniform flow compared to Sharp edge and Converging Inlets. The Bell mouth Inlet is the best suited inlet for the Duct with minimum pressure drop at 97 to 99mm radius. The best call will be a bell mouth inlet of 97.5mm radius (i.e radius (r) is 25% of duct diameter) whose pressure loss and inlet loss is minimum and the flow profile is uniform.

6. References

[1] Zachary M. Hall, Experimental Investigation of an Axisymmetric Turbofan Diffuser, M.Sc thesis, Auburn University, Alabama, 2009.

[2] Gerard Laruelle, Air intakes role, constraints and design, ICAS 2002 Congress, Research EADS Launch Vehicles, Les Mureaux, France, pp 643.1-643.16, 2002.

[3] Jack D. Mattingly, William H. Heiser, David T. Pratt, Aircraft engine design, AIAA Education Series, Reston, VA 20191-4344, August 2002.

[4] R. Allan Wallis, Axial flow fans and ducts, Wiley & Sons, Incorporated, John, University of California, Chapter 4, pp 61-136, 2009.

[5] Ashrae Handbook, Duct design, American Society of Heating, Refrigerating and Air-Conditioning Engineers, Chapter 35, pp 35.1-35.66, 2005.

[6] Mahendra M. Dhongadi, Distortions in Axial Flow Ducted Fan and Solving Techniques, Proceeding Of National Conference On "Innovations In Mechanical Engineering" at

Sinhgad Institute Of Technology, Lonavala, Pune, Maharashtra. 410 401 April 19th & 20th,2012.

[7] Mahendra M. Dhongadi,K.V. Sreenivas Rao,G.R. Srinivas, C.P.S. Prakash, Experimental Study of Fully Developed Flow in an Axial Ducted Fan Set Up, IJERT, Vol.2 Issue 6,pp685-690, June-2013.

[8] Mahendra M. Dhongadi,K.V. Sreenivas Rao,G.R. Srinivas, C.P.S. Prakash, Experimental Analysis of Unsteadiness at Inlet of Axial Fan Due to Stall and Surge, IJERT, Vol.2 Issue 6,pp718-723, June-2013.