STUDY ON CFD ANALYSIS OF SUBSONIC FLOW IN AFTER BURNER DIFFUSER DUCT

Surendra Kumar[1], Priyanka Jhavar[2]

[1]*Research Scholar, Mechanical Engineering, School of Engineering, SSSUTMS, MP, India*
[2]*Assistant Professor, Mechanical Engineering, School of Engineering, SSSUTMS, MP, India*

ABSTRACT

This paper present the study on CFD analysis of subsonic flow in after burner diffuser duct and it also discussed method to recover the residual flow swirl at the turbine exit, in order to ideally feed the afterburner core section with almost a no-swirl flow. Here we also study about the reduce flow velocity at the entry of afterburner combustion chamber, in order to make combustion in the core stream stables straighten the flow in order to obtain a flow ideally parallel to engine centerline, maximizing engine thrust. The proposed Design the model of airfoil struts in catia. The after burner diffuser duct is incorporated with airfoil struts. Discretized into Eight parts. To analyze the NACA 0012 symmetrical airfoil profile used in the design of struts.

Keyword: **-** *CFD Analysis 1, Combustion 2, Afterburner 3, and Swirl Flow 4.*

---

1. INTRODUCTION

A gas turbine afterburner is a thrust augmenter, which provides as on demand boost in thrust by re-burning the exhaust gas. The afterburner considerably raises exhaust gas temperature to increase the engine thrust. The primary combustor of the gas turbine engine only burns about 25 percent of the air. Thus, the afterburners can burn up to the remaining 75 percent of the initial air. Even though the afterburning is used for the short durations, the afterburner is permanently installed and it will impart total pressure losses to the flow even when not in use (called the dry condition) and thus will decrease the thrust and increase the specific fuel consumption (SFC) of an engine. The afterburner consists of exhaust diffuser, fuel injector, V-gutter as a flame stabilizer, liner with chute, anti-screech holes and cooling ring holes and nozzle. The aerodynamic characteristics of the diffuser between the turbine outlet and the afterburner inlet have an important bearing on the performance of the afterburner. This component, placed downstream of Low Pressure Turbine (LPT) exit, has different purposes.

- To recover the residual flow swirl at the turbine exit, in order to ideally feed the afterburner core section with almost a no-swirl flow.
- To reduce flow velocity at the entry of afterburner combustion chamber, in order to make combustion in the core stream stables straighten the flow in order to obtain a flow ideally parallel to engine centerline, maximizing engine thrust.

The overall geometry of the diffuser of the afterburner is basically dictated by the desired flow Mach number upstream of the flame stabilizer section. At the reheat design point, this Mach number has been selected in the 0.2-0.3 range. The calculation of the inlet-to-outlet area ratio of the diffuser is therefore straightforward on the basis of continuity equation. However the diffusion angle and length required for this area ratio have to be determined. In the exhaust diffuser of the afterburner, the outer wall of the diffuser is also the inner wall of the bypass duct, which is nearly straight. Therefore all the flow diffusion has to be obtained on the diffuser inner side.

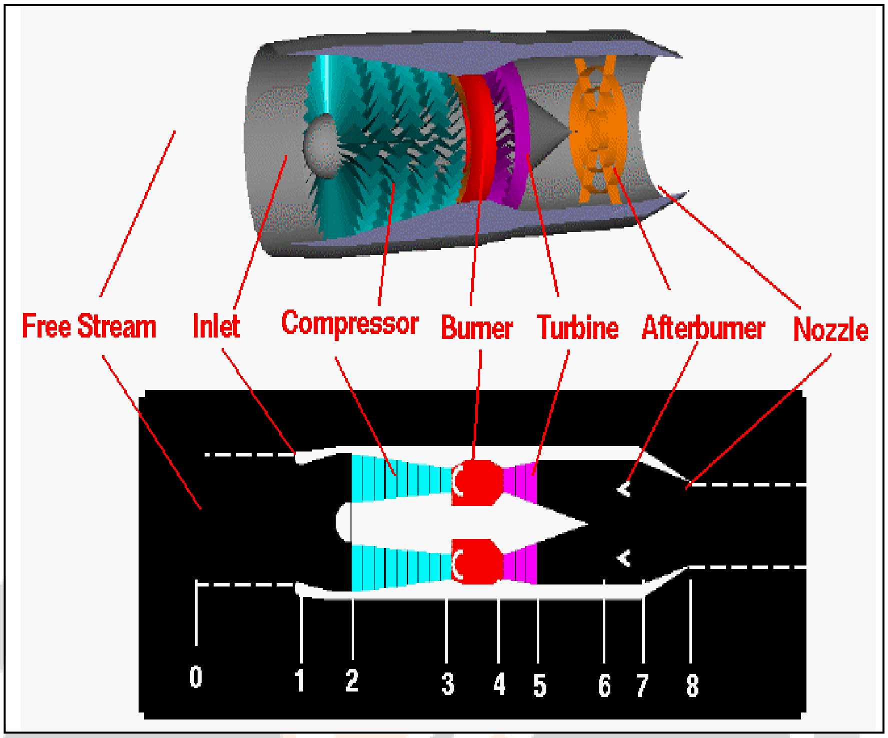

Fig 1. Schematic of a Turbojet Engine with after burner

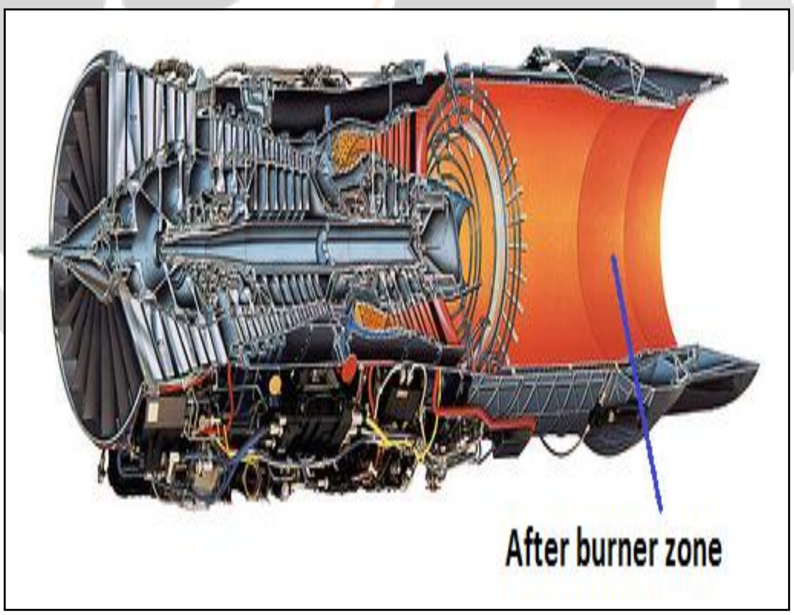

Fig 2. Sectional view of a Pratt & Whitney Turbojet Engine with after burner

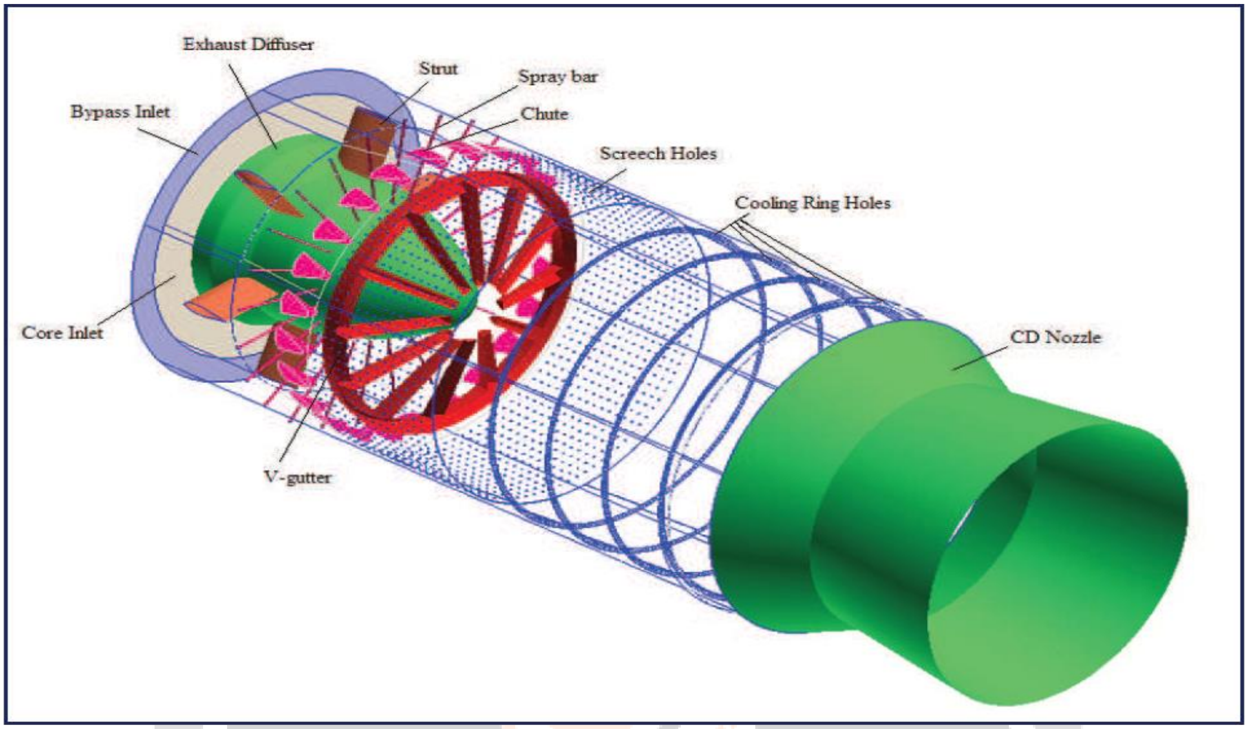

Fig 3.CAD model of the afterburner [2]

A gas turbine afterburner is a thrust augmenter, which provides as on demand boost in thrust by re-burning the exhaust gas. The afterburner considerably raises exhaust gas temperature to increase the engine thrust. The primary combustor of the gas turbine engine only burns about 25 percent of the air. Thus, the afterburners can burn up to the remaining 75 percent of the initial air. Even though the afterburning is used for the short durations, the afterburner is permanently installed and it will impart total pressure losses to the flow even when not in use (called the dry condition) and thus will decrease the thrust and increase the specific fuel consumption (SFC) of an engine. The afterburner consists of exhaust diffuser, fuel injector, V-gutters a flame stabilizer, liner with chute, anti-screech holes and cooling ring holes and nozzle. The aerodynamic characteristics of the diffuser between the turbine outlet and the afterburner inlet have an important bearing on the performance of the afterburner. This component, placed downstream of Low Pressure Turbine (LPT) exit, has different purposes.

• To recover the residual flow swirl at the turbine exit, in order to ideally feed the afterburner core section with almost a no-swirl flow.

• To reduce flow velocity at the entry of afterburner combustion chamber, in order to make combustion in the core stream stable.

• To straighten the flow in order to obtain a flow ideally parallel to engine centerline, maximizing engine thrust.

The overall geometry of the diffuser of the afterburner is basically dictated by the desired flow Mach number upstream of the flame stabilizer section. At the reheat design point, this Mach number has been selected in the 0.2-0.3 range. The calculation of the inlet-to-outlet area ratio of the diffuser is therefore straightforward on the basis of continuity equation. However the diffusion angle and length required for this area ratio have to be determined. In the exhaust diffuser of the afterburner, the outer wall of the diffuser is also the inner wall of the bypass duct, which is nearly straight. Therefore all the flow diffusion has to be obtained on the diffuser inner side. The objective of this work is to estimate the pressure loss contributed by the airfoil struts using FLUENT CFD SOFTWARE.

2. LITERATURE SURVEY

1 Dr. Mohammed sheriff et.al [1] conducted Experimental and Numerical investigation of complex flow in the after burner unit and emphasized on the deswirlingof flow in the afterburner diffuser duct portion due to the presence of airfoil struts. The study also reiterated the complexity of the flow due to the presence of complex geometrical shapes like V-gutter, fuel Injector and the combined influence on total pressure loss.

2. Yogesh.T.V, Dr.Ganesan.S and Dr.KishoreKumar.S [2] conducted numerical simulation of reacting turbulent flow in the after burner unit under development at Gas Turbine Research Establishment and emphasized on the need of flow deceleration and de-swirling in the diffuser duct from flame stability and combustion perspective.

3. PROBLEM STATEMENT

In the Aircraft Turbojet engine after the low pressure turbine, the flow gets swirled hence, in the after burner the velocity gets drastically increased. In the combustion chamber of the after burner the fuel and air is not get mixed properly hence the fuel is wasted. The swirled floe leads to loss of velocity in-turn reducing the efficiency and performance of the after burner.

4. SCOPE OF THE PRESENT STUDY

The present study focuses on studying the non-reacting, compressible flow in the after burner diffuser duct using FLUENT CFD software. The objective is analyze the flow de-swirling and pressure loss in the after burner diffuser duct due to the presence of airfoil struts.

5. METHODOLOGY

5.1 Phase-1

Aircraft Engine design book by Jack D. Mattingly presents the subsonic diffuser duct geometry along with the experimental data. The same diffuser duct is considered in this work.
The geometry of the diffuser duct is shown in Fig 1.The experimentally measured values are shown in Table1.

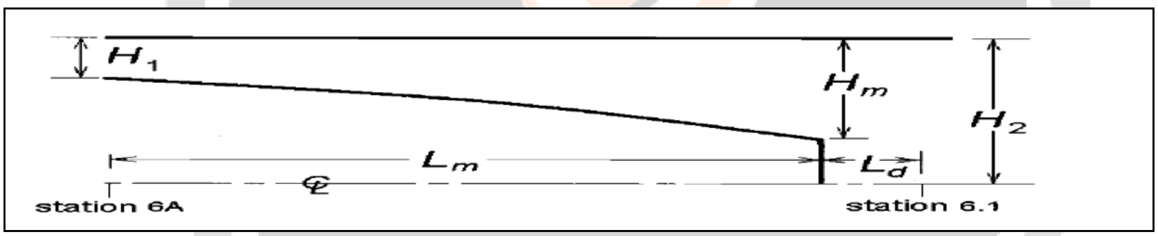

Fig 4.Geometry of the Diffuser duct (Reference, Aircraft Engine design, Jack D. Mattingly)

Table 1. Experimental results for afterburner diffuser geometry shown in Fig.1 (Reference, 03 Aircraft Engine design, Jack D. Mattingly)

Station	m dot (lbm/s)	gamma	Pt (psia)	Tt (R)	Press (psia)	Mach	Velocity (ft/s)	Area (ft^2)	Area* (ft^2)	I (lbf)
6A	228.63	1.3360	76.859	1513.71	68.248	0.4249	779.89	2.338	1.536	28516.0
6.1	**228.63**	**1.3360**	**76.181**	**1513.71**	**74.538**	**0.1817**	**268.19**	**5.378**	**1.645**	**60270.7**

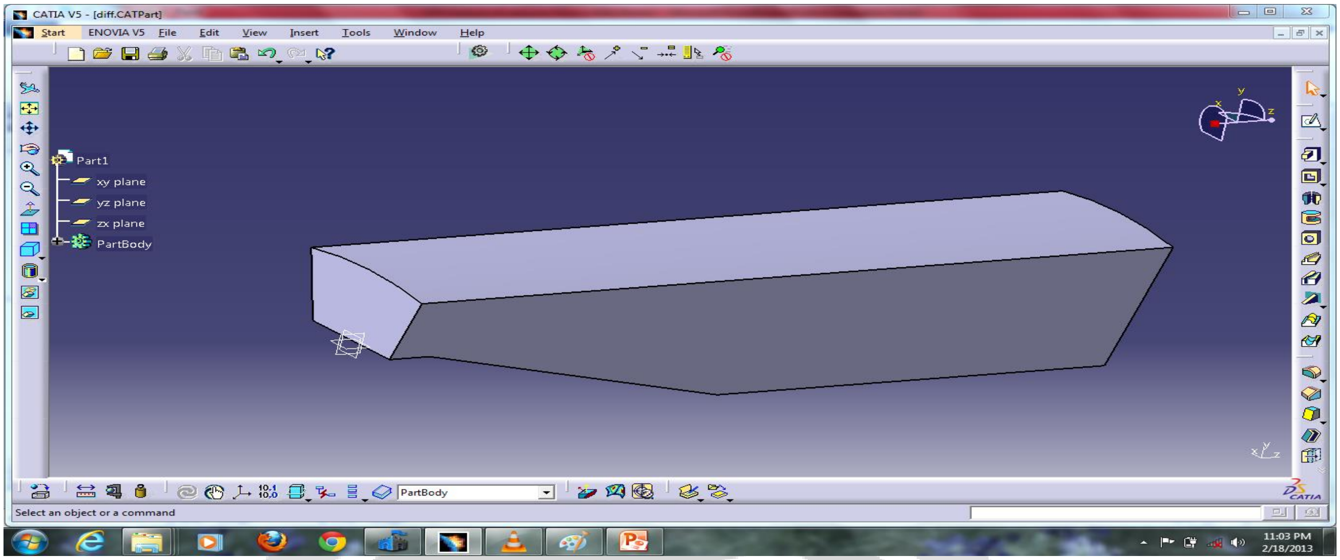

Fig 5: Modeling of diffuser as per experimental data

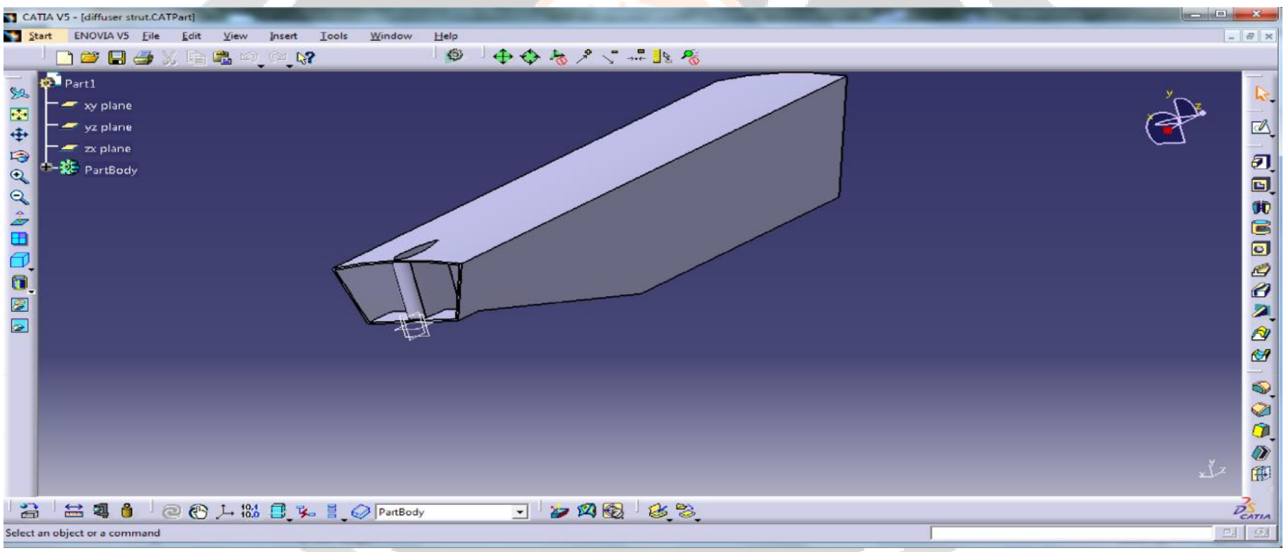

Fig 6: cutting out the airfoil strut

CFD analysis is carried out for the diffuser duct and analysis results are compared with the experimental data. This serves the validation of CFD analysis procedure.

5.2 Phase-2

In this phase, the after burner diffuser duct is incorporated with eight number of airfoil struts. NACA 0012 symmetrical airfoil profile [4] is used in the design of struts.

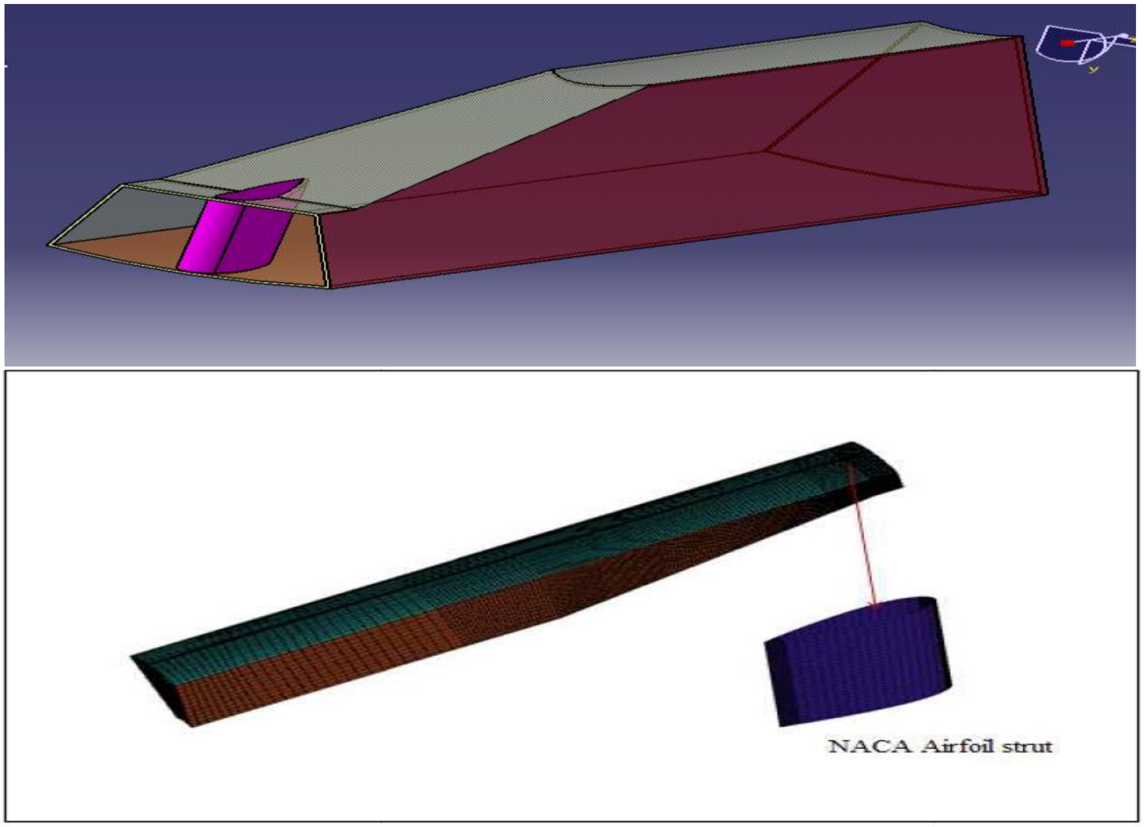

Fig 7: Front view and Side view of the Airfoil strut

CFD analysis is carried out for the after burner diffuser duct with airfoil struts and the contribution of struts for total pressure loss is analyzed using CFD analysis. The flow de-swirling due to the struts is also studied.

6. GEOMETRIC MODELING AND GRID GENERATION

For any flow problem it is important to describe the physical boundaries that contain the fluid as well as the barriers over which the fluid flow has to take place. Geometric modeling takes care of the above aspects by the generation of a computational model. Various sources of geometrical data are available for describing the geometry of bounding surfaces and obstructions e.g. engineering drawings and databases created by computer-aided design systems. Modeling effort depends on the complexity of the flow domain and it is often helpful to simplify the geometry wherever possible without sacrificing the accuracy of the simulation.

Grid generation is the next step in the process of CFD simulation and involves the sub-division of the domain into a number of smaller, non-overlapping sub-domains often referred as a grid (or mesh) of cells (or control volumes or elements) depending upon the type of numerical discretization technique used. The number of cells in the grid governs the accuracy of a CFD solution. Finer the grid, better the accuracy. Both the accuracy and computational cost in terms of computer hardware and calculation time are dependent on the fineness of the grid. Optimal meshes are often non-uniform with fine meshing in critical regions of the domain where large gradients in flow variables are expected and coarser mesh in regions where the variations are relatively small. The arrangement of cells in the grid also called as topology of the grid can be of two types, structured or unstructured. In a structured grid the cells are well ordered and a simple scheme (e.g., j, k indices) can be used to label the elements and identify neighbors. Structured grids come in several varieties depending on the shape of their cells. The simplest grid is generated from rectangular brick cells though their use is limited by the fact that geometric surfaces are usually approximated by blocking out entire cells, which leads to boundaries having discrete steps thereby introducing undesirable effects. Better geometric representations of curved obstacle surfaces can be achieved either by deforming the grid elements to conform with specified geometric shapes, the resulting cells then have general hexahedral shapes and the grid is often referred to as a body-fitted grid or by retaining the rectangular cells but supplementing them with some means of defining obstacles cutting through their interiors.

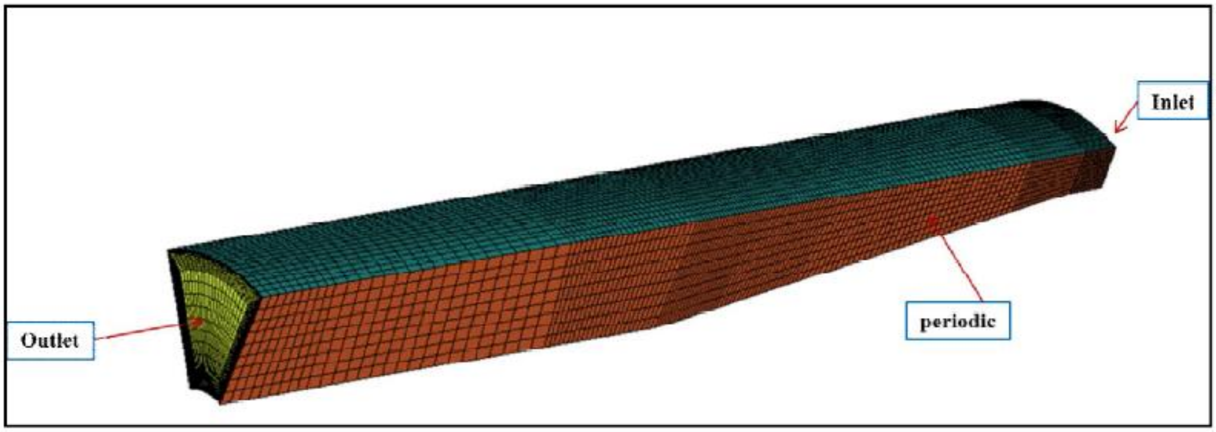

Fig 8. Meshing the model

In unstructured grids the cells can be joined in any manner and special connectivity lists must be kept for identity in neighboring cells. Unstructured grids have the advantage of generality in that they can be made to conform to nearly any desired geometry. However the grid generation process is not completely automatic and may require considerable user interaction to produce grids with acceptable degrees of local resolution while at the same time having a minimum of element distortion.

Unstructured grids require more information to be stored and recovered than structured grids and changing element types and sizes can increase numerical approximation errors. A popular type of unstructured grid consists of tetrahedral elements. These grids tend to be easier to generate than those composed of hexahedral elements, but they generally have poorer numerical accuracy.

In summary, the best choice for a grid system depends on several factors viz, Convenience in generation, memory requirements, numerical accuracy, flexibility to conform to complex geometries and flexibility for localized regions of high or low resolution.

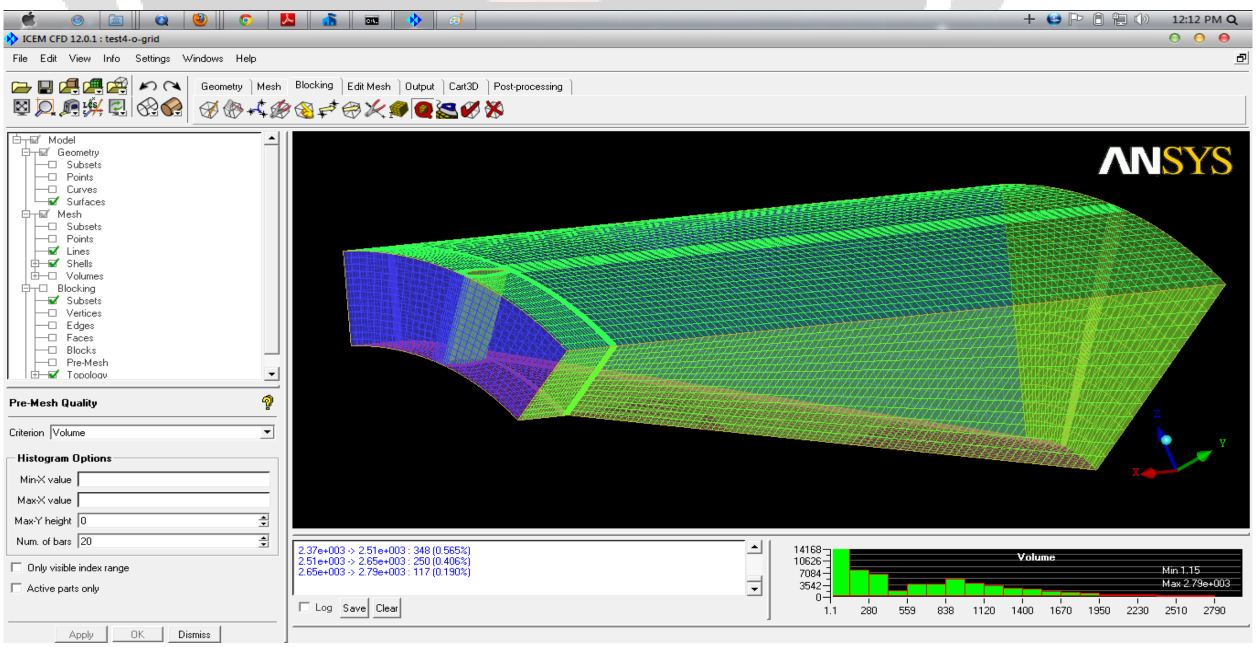

Fig 9. Meshing the Model in ANSYS

6.1 Flow Specification

The specification of the flow problem tells the CFD software the exact problem that is to be solved and it is achieved by performing the following tasks:

6.1.1 Specifying Fluid Properties

Fluids possess a variety of properties and the solver program must be given some way of calculating the values of these. Generally depending on the flow problem the values are given as constants or a relationship with other independent variables is given or a standard variation of the property is chosen.

6.1.1.1 Determining he flow Variables to Be Calculated

The variables that are needed depend on the way in which the governing equations have been discretized and the algorithm set up to solve them. Further, the nature of the flow and the type of models used for modeling turbulence, heat transfer etc. dictate the selection of the variables.

6.1.1.2 Defining the Boundary Conditions

Proper Prescription of the boundary conditions and their modeling are the most crucial factors that influence the computed results. The form of the boundary conditions that is required by any partial differential equation depends on the equation itself and the way in which it is has been discretized. Some common boundary conditions are, however, met when solving fluid flow problems with CFD and are as follows.

- **Inlet:** At inlet to the domain, velocity, pressure, mass flow can be specified. Also, the turbulence variables such as k and ε can be specified. Inlet boundaries require the specification of the distribution of all the floe variables.
- **Outlet:** This marks the exit of the domain. Normally, the gauge pressure is set to Zero at the outlet. The velocity components and turbulence variables will have a zero spatial derivatives in a direction normal to the exit boundary.
- **Symmetry:** When the flow is symmetrical about some plane there is no flow through the boundary and the derivatives of the variables normal to the boundary are set to be zero.
- **Wall:** It is the most common boundary encountered in confined fluid flow problems. The no-slip boundary condition is enforced at the wall for viscous flow. The shear stress and the heat transfer between the fluid and wall are computed based on the flow details in the local flow field. In turbulent flows the near wall region is mostly modeled using semi-empirical formulas called “wall functions”. These functions bridge the viscosity-affected region between the wall and the fully turbulent flow region.

6.1.1.3 Cyclic or periodic boundaries:

These boundaries come in pairs and are used to specify that the flow has the same values of variables at equivalent positions on both of the boundaries.

- **Defining Initial conditions:** Many solution algorithms require that some form of initial flow filed be specified to the solver. This could be due to the flow being time dependant, where the initial state of the variables is required to start the calculation, or due to the use of a quasi-time-varying solution algorithm. Equally, the non-linearity of the problem will demand some initial guess for the variables, which need to be supplied as a set of default values or by the user. If turbulence variables are being used then they are usually set to a small positive value or some realistic value.

6.2 Solution Procedure (Versteeg and Malalsekera 1995)

There are several numerical algorithms for solving the discrete equations, among which SIMPLE algorithm for correct linkage between pressure and velocity and the TDMA line-by-line solver of the algebraic equations are the most popular. The success of the numerical solution algorithm is determined from the mathematical concepts of convergence and stability. In the following sections some of the terminology and techniques associated with a numerical solution are discussed.

6.2.1 Convergence, consistency and stability

- **Convergence:** It is a property of the numerical algorithm to produce a solution, which approaches the exact (analytical) solution if such a solution exists, as the grid spacing control volume size or element size is reduced to zero.
- **Consistency:** It is the ability of the numerical scheme to produce systems of algebraic equations, which can be demonstrated to be equivalent to the original governing equation, as the grid spacing tends to zero.
- **Stability:** It is associated with damping of errors as the numerical method proceeds. A process is stable if the equations move towards a converged solution such that the errors in discrete solution do not swamp the results by growing as the numerical process proceeds.

6.3 Solution Algorithm for Pressure-Velocity Coupling

The continuity and momentum equations, which govern the flow, are intricately coupled because every velocity component appears in each momentum and continuity equations. The most complex issue to resolve is the role played by the pressure. Pressure appears in all the momentum equations but there is evidently no equation for pressure. More over the non-linear quantities in the convective terms of the momentum equations pose additional problems for obtaining solution to the equation set. The problems associated with the non-linearity of the momentum equations and coupling between the transport equations are tackled by adoptive iterative solution strategy such as SIMPLE (semi-Implicit Method for Pressure-linked Equations) algorithm. In this iterative method when other scalars are coupled to the momentum equations, the calculations are done sequentially. Starting from an initial pressure field its principal steps are as follows:

- Solving the discretized momentum equations to yield the intermediate Velocity field.
- Solving the continuity equation in the form of an equation for pressure correction.
- Correction of pressure and Velocity.
- Solving other discretized transport equations for scalars.
- Repetition of the above process till convergence is achieved.

Refinements to SIMPLE have produced more economical and stable iteration methods like SIMPLER (SIMPLE-Revised) and SIMPLEC (SIMPLE-Consistent), PISO algorithm, which stands for Pressure Implicit with Splitting of Operators, contains an additional correction step to SIMPLE to enhance its performance per iteration. SIMPLEC and PISO have proved to be as efficient as SIMPLER in certain types of flows but it is not clear whether it can be categorically stated that they are better than SIMPLER. Comparisons have shown that the performance of each algorithm depend on the flow conditions, the degree of coupling and between momentum and scalar equations and on the amount of under-relaxation used.

6.4 Solution of Discretized Equations

The system of linear algebraic equations obtained from the discretization of governing equations is solved either by direct methods or iterative methods. Cramer's rule matrix inversion and Gaussian elimination fall under the category of direct methods while Jacobi and Gauss-Seidel methods are well known examples of iterative methods. Iterative methods are based on repeated application of relatively simple algorithm leading to eventual convergence. The main advantage of iterative methods over the direct methods is that only non-zero coefficient of equations need to be stored in core memory.
Jacobi and Gauss-Seidel methods though quite easy to implement in simple computer programs, they can be slow in convergence when the system of equations is large as often the case in CFD. Tri-diagonal matrix algorithm (TDMA), through actually a direct method for one-dimensional situations is being widely used in CFD programs to solve multi-dimensional problems by applying iteratively in line-by-line on a selected plane and the proceeding the calculation to next plane, scanning the domain plane by plane. This method is highly economical for tri-diagonal system. However if discretization schemes are used that incorporate influences from location other than the immediate neighbors or if body-fitted co-ordinate system are used then it may be necessary to resort to alternative techniques such as penta0diagonal matrix algorithm or stone's implicit procedure and conjugate gradient method.

6.5 Controlling the Iterative Process

To avoid the escalation of the residual errors from iteration to iteration and eventually leading to the divergence of solution we need to control the process. Under-relaxation factors are usually employed to take care of the controlling aspect. These take the solution calculated during the current iteration and scale it so that the solution used in the next iteration is not too different from the solution at start of the current iteration.
This is mathematically represented as:

$$\phi_{new} = \alpha\phi_{cal} + (1-\alpha)\phi_{old}$$

ϕ_{old} is the value of the variable at the start of the document iteration while ϕ_{cal} is the value at the end of the iteration.

ϕ_{new} is produced after process of scaling by the relaxation factor α whose value lies between zero and one. Another means of controlling the overall solution process is to use a time dependent solution scheme, even if the flow is known to be steady. With time dependent scheme the main controlling factor is the value of the time step. This is to give as small number of time steps as possible whilst maintaining a smoothly converging solution.

6.5.1 Convergence Criteria

Final convergence is decided by the way of residual source criterion. The convergence criterion in the present study is set as 10^{-6} for all the parameters.

6.6 Fluent Package

FLUENT is a state-of-the-art computer program for modeling the fluid flow and heat transfer in complex geometries. It provides comprehensive modeling capabilities for wide range of incompressible and compressible, laminar and turbulent fluid flow problems. Further, a broad range of mathematical models for transport phenomena (heat transfer and chemical reactions) is combined with the ability to model complex geometries. FLUENT uses unstructured meshes in order to reduce the amount of time spent on grid generation, simply the geometry modeling and facilitate modeling more complex geometries than those which can be handled with conventional multi-block structured meshes. FLUENT can also use body-fitted and block-structured meshes. Moreover the solution-adaptive grid capability provides means for accurately predicting flow fields with large gradients. This feature also reduces the computational effort required to achieve a desired level of accuracy.
FLUENT is written in the C language and makes full use of the flexibility and power offered by the language. In addition, FLUENT uses a client/server architecture, which allows it to run as separate simultaneous processes on client desktop workstation and powerful computer servers. All functions required to compute a solution and display the results accessible in FLUENT through an interactive, menu-driven interface which is written in a language called Scheme, a variant of LISP.

7. CONCLUSION

This paper presents the study of study on CFD analysis of subsonic flow in after burner diffuser duct. Here our study is focused on the combustion chamber of the after burner the fuel and air is not getting mixed properly hence the fuel is wasted. The swirled flow leads to loss of velocity in-turn reducing the efficiency and performance of the after burner. The present study focuses on studying the non-reacting, compressible flow in the after burner diffuser duct using FLUENT CFD software. The objective is analyze the flow de-swirling and pressure loss in the after burner diffuser duct due to the presence of airfoil struts.

REFERENCE

- Dr.N.Mohammed Sheriff et. al, “CFD analysis of flow in Afterburner”, Proceedings of the 6th WSEAS International Conference on HEAT and MASS TRANSFER (HMT'09).
- Yogesh TVet.al “Effect of Exhaust Diffuser on Gas Turbine After Burner Diffuser performance” ,Proceedings of the 37th National & 4th International Conference on Fluid Mechanics and Fluid Power December 16-18, 2010, IIT Madras, Chennai, India..
- Jack D. Mattingley et.al, “Aircraft Engine Design” AIAA Education series, 2001.
- Abbott, “Theory of Wing sections “McGraw Hill, 1949.
- S. M. Yahya, “Fundamentals of Compressible flow “New Age International Publishers, 2000.
- Dr. Isaac, J.J, Rajashekar. C, N.R., Ramesh, et al, Afterburner flow visualization studies in a water tunnel, NCABE-paper-1992.
- Mattingly J.D, et al. Aircraft engine design, AIAA Educational series, and 1988.
- Philip P. Walsh, et al .Gas Turbine Performance, 2nd edition 2004.
- Bheemaraddi. S.B. Dr. S. Kumarappa, Assessment of Turbulent Boundary Layer Modeling Methods by Using Computational Fluid Dynamics for Gas Turbine Engine Afterburner Diffuser, Vol. 3, Issue 1, January 2014.
- K. M. Pandey, B. K. Azad, S. P. Sahu and M. Prajapati, Computational Analysis of Mixing in Strut Based Combustion at Air Inlet Mach number 2, Vol.2, No.1, February 2011.
- Dr. N. Mohmed Sheriff, P. Selva Kumar, CFD Analysis of Flow in After Burner.
- N Maheswara Reddy, E G Tulapurkara, V Ganesan, Optimization of the Fuel Manifold Location inside a Jet Engine Afterburner, December, 2010.
- S. Roga, K.M. Pandey, A.P. Singh, Computational Analysis of Supersonic Combustion Using Wedge-Shaped Strut Injector with Turbulent Non-Premixed Combustion.