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 pressure loss in the after burner diffuser duct without the presence of airfoil struts. Analyze the pressure loss in the after burner diffuser duct with the presence of airfoil struts. Mathematical methods like Navier stoke Equation and K-Epsilon Modeling Equation. In this paper 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. 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.

Keyword: **-** *FLUENT 1, CAD Model 2, Computational Domain 3, CFD Analysis 4, K-Epsilon Turbulence Model5.*

1. INTRODUCTION

In order to achieve better take-off characteristics, higher rate of climb and meet performance demands during tactical maneuvers, the thrust of an aero engine has to be augmented by employing an afterburner. The main advantages of using the afterburning gas turbine cycle is that the weight and size of the augmented engine are much less than that of a turbojet engine which can produce the same maximum thrust periodically.

Afterburning consists of the introduction and burning of kerosene fuel between the engine turbine and the jet propelling nozzle. Due to structural limitations, the maximum gas temperatures approaching the turbine are limited to around half the adiabatic flame temperature. Consequently, the gas leaving the turbine will still contain a considerable proportion of oxygen. Secondary burning of fuel in the afterburner leads to increased exit velocity and thrust.

Here paper discussed about the simulated results employing FLUENT helps in understanding the physics of fluid flow. As the simulations are based on nearly universal principles, modeling of a particular phenomenon will be reasonably complete and accurate. It involves less number of assumptions and empirical inputs. In the present study FLUENT code is suitably adjust to predict the flow domain of interest.

2. DESIGNING CRITERIA AND CAD MODELS

The CAD model is generated using CATIA V5 CAD package. Structured mesh is generated using commercial meshing software and details of the mesh are given below.

Owing to the periodicity of the geometry and physics, a 45 degree sector of the diffuser is considered for the CFD analysis. The diffuser with airfoil struts is also modeled as periodic with 45 degree sector, so that one airfoil strut is considered for the CFD analysis.

`

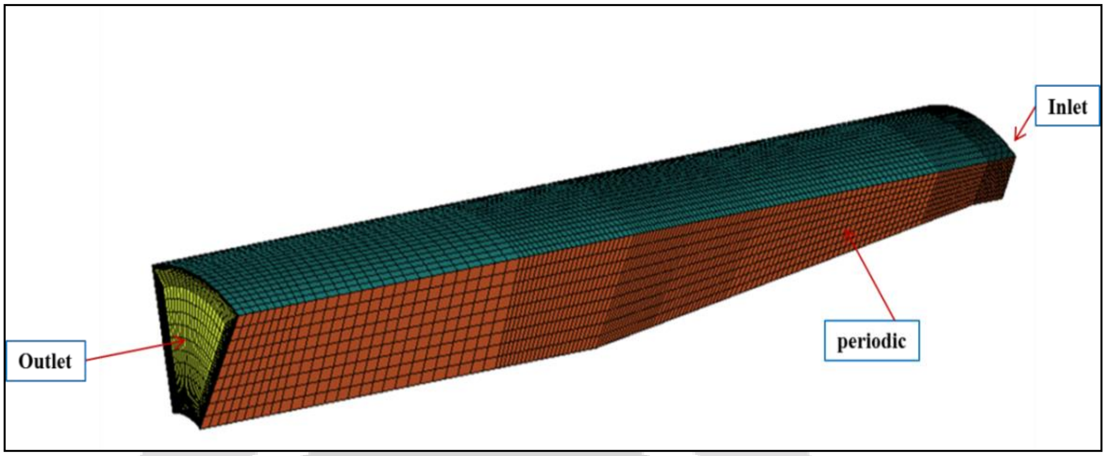

Fig 1. Computational domain of the after burner diffuser duct without struts(45 Degree Sector)

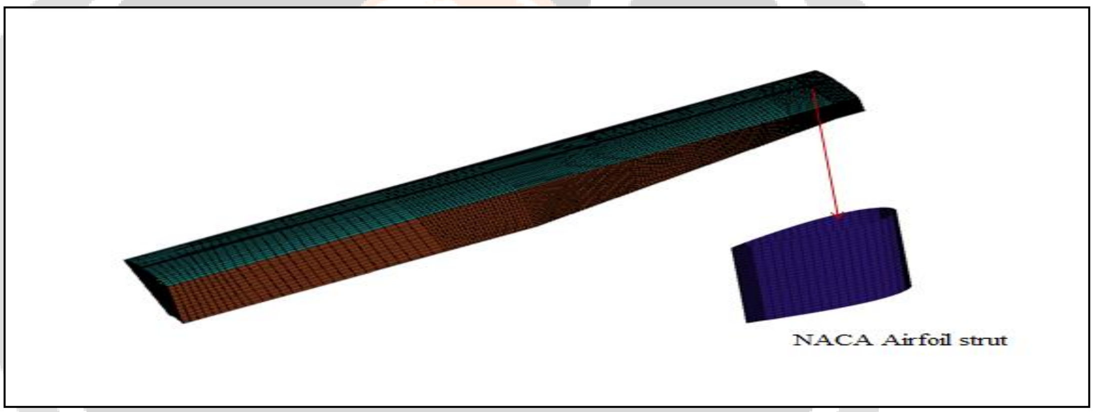

Fig 2. Computational domain of the after burner diffuser duct with struts(45 Degree Sector)

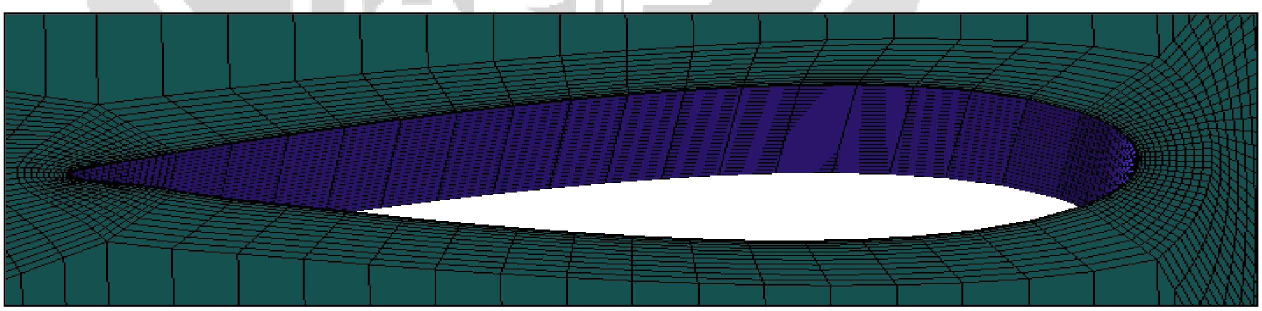

Fig 3. Body-fitted grid generated around the Airfoil Strut

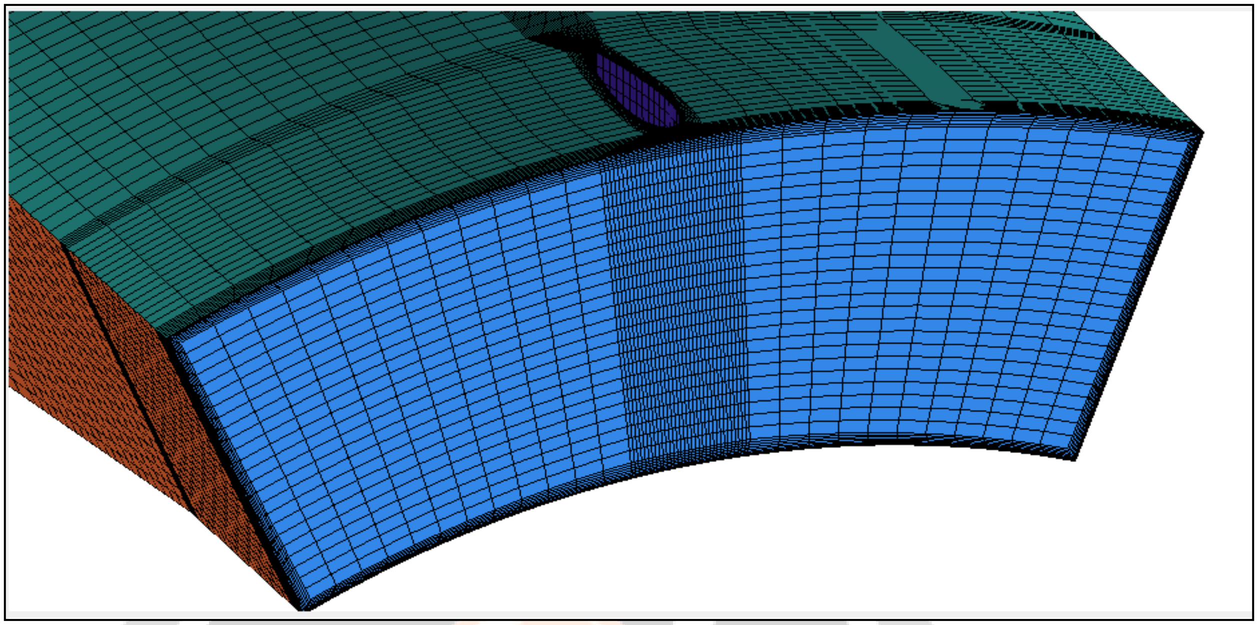

Fig 4. Mesh resolved near the wall region and around the airfoil to capture the viscous effects

The computational domain is discretized with hexahedral elements. Body fitted O-grid is generated to capture the viscous effects around the airfoil strut as well as near the wall regions as shown in Fig 3 and Fig 4.
The computational domain is discretized with hexahedral elements. Body fitted O-grid is generated to capture the viscous effects around the airfoil strut as well as near the wall regions as shown in Fig 7 and Fig 8.
The final mesh for the afterburner diffuser duct without struts consists of around 150000 hexahedral elements and for the afterburner diffuser duct with airfoil struts the mesh consists of around 280000 hexahedral elements.
The final mesh size is arrived by conducting grid independence studies, which is explained in Table.02

3. BOUNDARY CONDITIONS

The following boundary conditions have been imposed for CFD analysis,
A. Inlet: Absolute Total pressure of 529924 Pa and Total temperature of 686.5 Degree Kelvin is imposed at the inlet boundary.
B. Outlet: Mass flow value of 12.96 Kg/sec corresponding to 45 degree sector is imposed at the outlet.
C. Wall: The walls of the diffuser duct, airfoil strut surfaces are imposed with no-slip boundary condition.
D. Periodicity: Rotational periodicity is imposed on both the periodic surfaces, shown in Figure.05.

3.1 Fluid Cells

The fluid cells /fluid volumes are imposed with air material properties. Ideal gas equation is used to calculate the density as a function of pressure and temperature, which is characteristic of compressible flows.

4. RESULTS-DISCUSSIONS

A. Grid Independence Studies

The gird independence studies are carried out to finalize the mesh which captures the results accurately. Table 2 shows the grid independence study details.

Table. 1. Grid Independence study results

S l No	Grid size, Number of elements	Y-plus	Total pressure (absolute)		Remarks
			Experimental	CFD	
Afterburner diffuser duct without struts					
1	80031	78	525249	503689	Y-plus value is high
2	135268	38	525249	521963	Y-plus value is acceptable
3	157575	32	525249	525003	Mesh finalized for CFD analysis

Table.2 Afterburner diffuser duct with NACA0012 airfoil Strut

Afterburner diffuser duct with NACA 0012 airfoil struts

4	219713	74	Not available	510697	Y-plus value is high
5	259871	35	Not available	521321	Y-plus value is acceptable
6	286961	34	Not available	522632	Mesh finalized for CFD analysis

- The standard wall function approach in K-epsilon Turbulence model requires a Y-plus value ranging between 30 to 300.

The standard wall function approach in K-epsilon Turbulence model requires a Y-plus value ranging between 30 to 300, corresponding to the Log-Law region of the turbulent boundary layer. However Y-plus value around 35 to 40 will give the best results.

From the above table it is concluded that the final mesh selected for CFD analysis is appropriate.

B. **CFD Analysis Results**

Table 03.CFD analysis results for the afterburner diffuser duct without struts

Sl No	Description	Unit	Values at station 6.1	
			Experimental	CFD
1	Mass flow rate	Kg/sec	12.9642	12.9642
2	Absolute Total pressure	Pascal	525249	525003
3	Absolute static pressure	Pascal	513920	512809
4	Total temperature	K	686.5	686.50
5	Mach Number		0.1817	0.1642

Table 04.CFD analysis results for the afterburner diffuser duct with struts

Sl No	Description	Unit	Values at station 6.1
			CFD
1	Mass flow rate	Kg/sec	12.9642
2	Absolute Total pressure	Pascals	522632
3	Absolute static pressure	Pascals	518587
4	Total temperature	K	686.498
5	Mach Number		0.164526

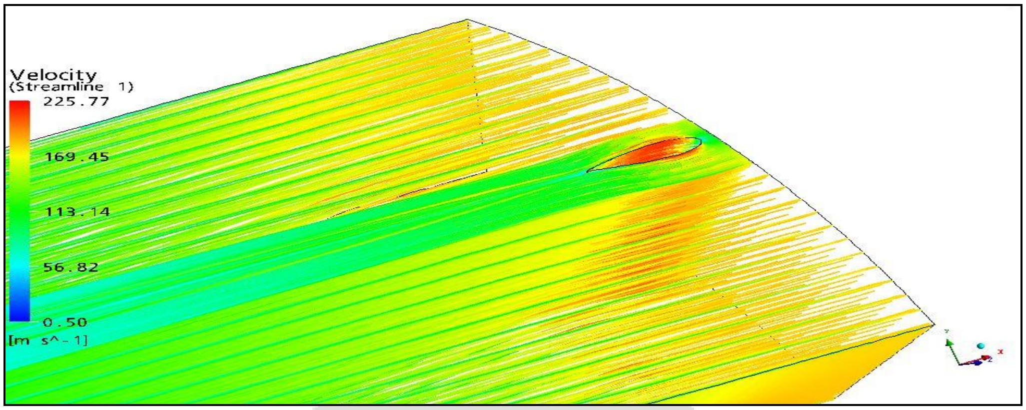

Fig 5.Stream lines around the airfoil strut

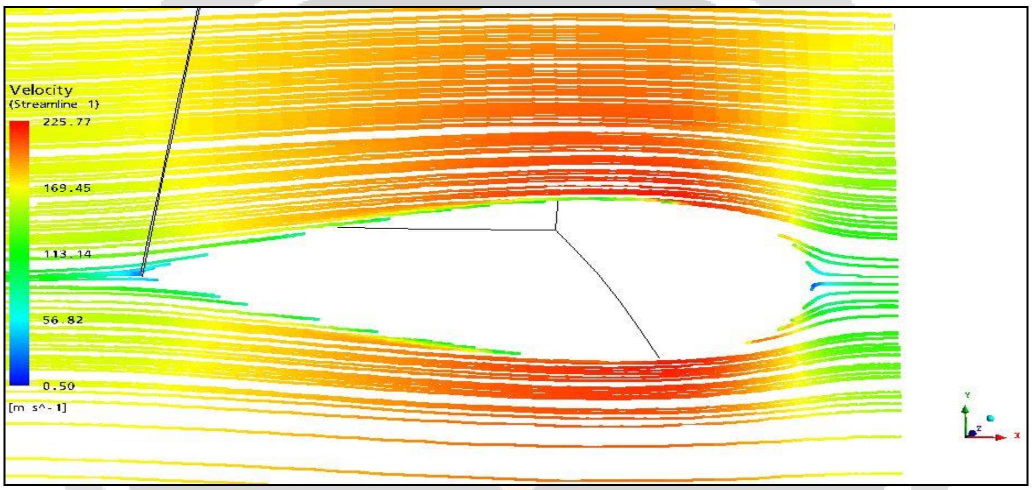

Fig 6. Stream lines around the airfoil strut (Sectional view)

Table 3, shows CFD analysis results for after burner diffuser duct without struts. The analysis results are compared with the experimental data available in the literature (Aircraft Engine design, Jack D. Mattingly).
Table.3 indicates that the experimentally measured total pressure loss between station 6A (Inlet) and Station 6.1 is 4675 Pascal's, while that predicted by CFD analysis is 4921 Pascals.
Table 4, Shows CFD analysis results for afterburner diffuser duct with NACA 0012 airfoil struts. The total pressure loss with struts is 7292 Pascals. This higher value of pressure drop is due to the skin friction of the strut surfaces. Airfoil struts contribute to an additional pressure loss of 2617 Pascals. An additional pressure loss of 2617 Pascals is practically low because the flow around the strut is attached to the strut surface and there is no flow separation from the strut wall. It is observed that the presence of struts increases the Turbulent Kinetic Energy (nearly two times) of the flow in the diffuser duct. This is desirable because increased turbulence leads to the better mixing of Fuel (which is injected to the after burner unit) with the air after station.

5. CONCLUSION

CFD analysis is carried out for the after burner diffuser duct with and without struts**.** CFD analysis results for the after burner diffuser duct without struts is compared with the experimental data. The contribution of NACA 0012 airfoil struts for pressure loss is estimated. Increase of Turbulence in the flow with struts is beneficial as it leads to better mixing of air and fuel in the after burner unit. It is observed that the presence of struts increases the Turbulent Kinetic Energy (nearly two times) of the flow in the diffuser duct. This is desirable because increased turbulence leads to the better mixing of Fuel (which is injected to the after burner unit) with the air after station.

REFERENCES

- 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.
- SM.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.Mohammed Sheriff, P.Selva Kumar, CFD Analysis of Flow in After Burner.
- N Maheswara Reddy, E G Tulapurkara, VGanesan, 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