e-ISSN: 2582-5208
International Research Journal of Modernization in Engineering Technology and Science
(Peer-Reviewed, Open Access, Fully Refereed International Journal)
Volume:06/Issue:04/April-2024 Impact Factor- 7.868 www.irjmets.com

ANALYSIS OF SUBSONIC FLOW IN AFTER BURNER DIFFUSER DUCT OF GAS TURBINE ENGINES USING CFD APPROACH

Chandra Shekhar Verma*1, Shailendra Kumar Bohidar*2

*1M.Tech Research Scholar, Dept. of Mech. Engg., MATS School of Engineering & IT, MATS University, Raipur, C.G., India.

*2Associate Professor, Dept. of Mech. Engg., MATS School of Engineering & IT, MATS University, Raipur, C.G., India.

ABSTRACT

Afterburner is an additional component of Turbojet engine employed in military aircrafts to provide an increasing thrust during transonic flight, takeoff and combat situations. Afterburning is achieved by injecting additional fuel in to the jet pipe downstream of the turbine. The subsonic flow coming out from the low pressure turbine is decelerated in the diffuser to provide a velocity low enough for satisfactory combustion. The diffuser is fitted with radial airfoil struts which deswirl the flow. The presence of Diffuser struts, Fuel injection rings, Flame holders makes the flow in the after burner more complex. The detailed study of flow is required for the design and performance evaluation of the after burner.

In this work the hot flow in the conceptual afterburner diffuser duct fitted with NACA 0012 airfoil struts is analyzed using Computational Fluid Dynamics method.

Further CFD analysis is carried out for the diffuser duct incorporated with NACA 0012 Airfoil struts and the behavior of the flow due to the presence of airfoil struts is analyzed. The swirl angle of the flow at the exit section of the diffuser, pressure loss due to the presence of airfoil struts is studied with CFD analysis results.

Keywords: NACA, CFD, Turbine, Combustor, Duct.

I. INTRODUCTION

1.1 ABOUT GAS TURBINE ENGINE

During World War II, a new type of aircraft engine was developed independently in England by Sir Frank Whittle and in Germany by Hans von Ohain and Max Hahn. Most modern passenger as well as military aircraft is powered by gas turbine engines. A simple gas turbine consist of three main components: compressor, combustion chamber and turbine and the ideal thermodynamic cycle for a simple gas turbine is the Brayton cycle, which consists of isentropic compression, constant pressure heat addition and isentropic expansion. Modern gas turbine engines come in a wide variety of shapes and sizes because of the many different aircraft mission profiles. In *turboprop* engines, by the combination of a propeller and gas turbine, optimum propulsive efficiency is achieved for low-speed aircrafts and helicopters. *Turbojet* engines produce thrust primarily from the direct impulse of exhaust gases; turbojet engines along with an afterburner are used in supersonic aircraft. For example, the Rolls-Royce Olympus turbojet engine was used to power the Concorde. *Turbofan* engines generate thrust from a combination of a high bypass ratio ducted fan as well as engine jet thrust; these engines are ideal for high subsonic aircraft speeds; most medium and long haul commercial aircraft use turbofan engines.

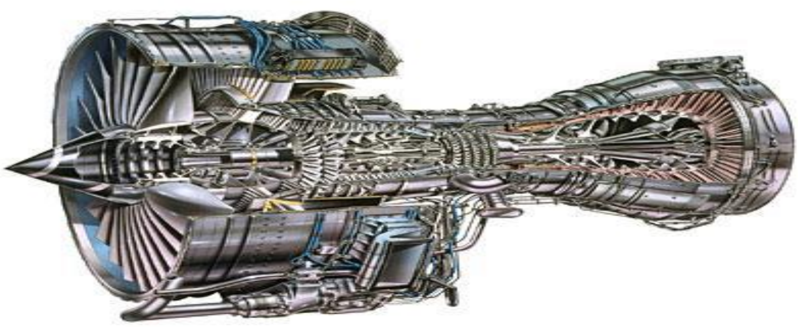


Fig 1: Rolls-Royce Trent 500 – Turbofan engine (www.rolls-royce.com)

1.2 Gas Turbine Afterburner

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.

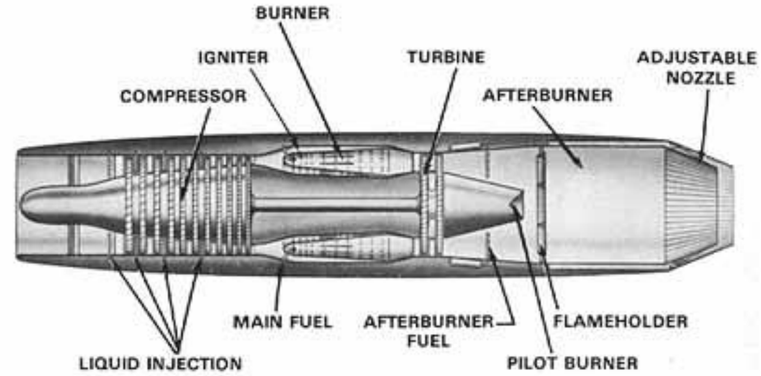


Fig 2: Gas turbine assembly with afterburner

This component, placed downstream of Low Pressure Turbine (LPT) exit, has different purposes.

1. 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.
2. To reduce flow velocity at the entry of afterburner combustion chamber, in order to make combustion in the core stream stable.
3. 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

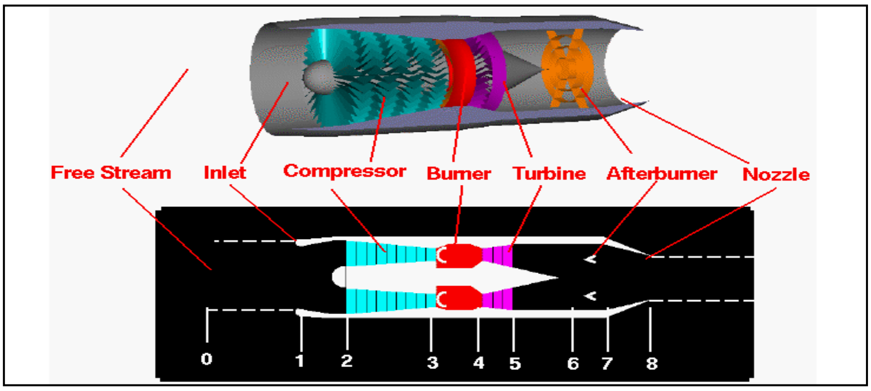


Fig 3: Schematic of a Turbojet Engine with after burner

e-ISSN: 2582-5208
International Research Journal of Modernization in Engineering Technology and Science
(Peer-Reviewed, Open Access, Fully Refereed International Journal)
Volume:06/Issue:04/April-2024 Impact Factor- 7.868 www.irjmets.com

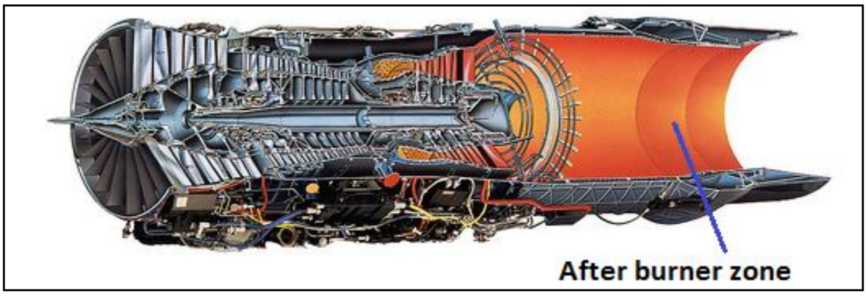


Fig 4: Sectional view off a Pratt & Whitney Turbojet Engine with after burner

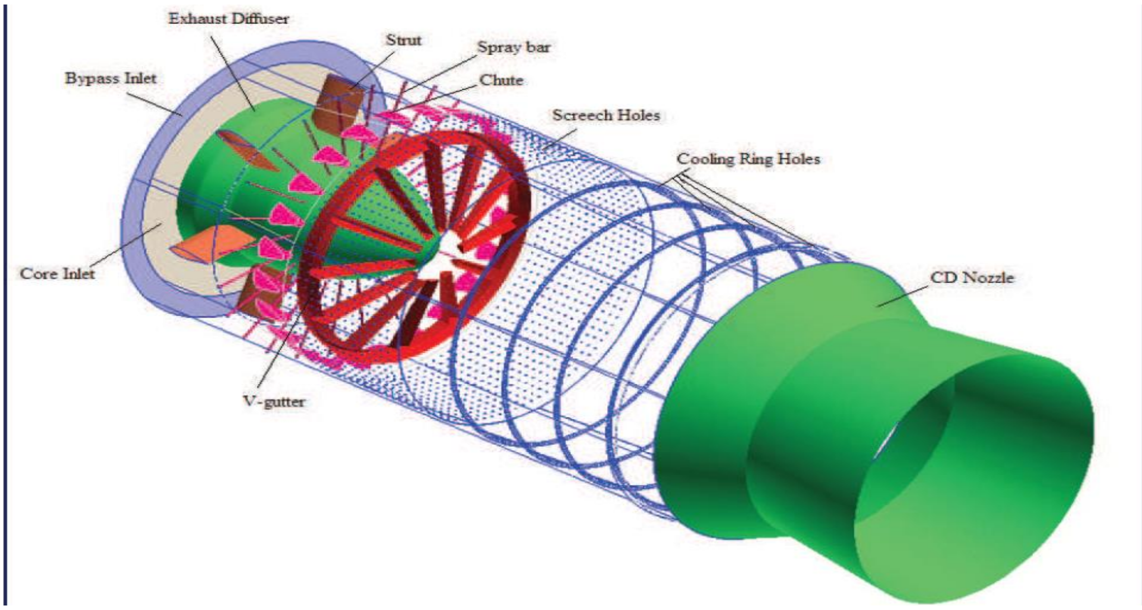


Fig 5: CAD model of the afterburner (2)

II. LITERATURE SURVEY

❖ **Ameri, M. and Dorcheh, F. J[2].**

The author has described the CFD modeling of heat recovery steam generator inlet duct the abilities of computational fluid dynamics have been assessed to obtain the crucial profiles without the experimental difficulties. Regarding the special characteristics of flow and geometry, numerical solution may not be performed without taking some techniques into the CFD modeling. The actual HRSG inlet channel incorporates one perforated plate to correct the flow and three burner elements inside its wide-angle diffuser. Investigations have shown that the perforated plate and heat exchanger modules can be modeled by porous jump boundary condition and the burner elements by radiator faces respectively. Realizable k-ε with non-equilibrium wall function seems to be the most optimum turbulence model for solution of the problem. The inlet duct flow field can be obtained numerically for the actual flow Reynolds number by removing the main stack, incorporating heat exchangers, swirl velocity and high inlet actual temperature. Realizable k-ε with non-equilibrium wall function can bean suggested as optimum turbulence model to estimate the HRSG inlet duct flow field. Results of numerical solution of HRSG inlet duct have shown that three large baffles destroy the exit velocity uniformity which was produced by suitable perforated plate.

❖ **Balogun, O., et al[3]**

The author has presented exergy analysis of gas turbine-burner engine, the exergy of the conventional and the turbine-burner engines was completed based on ideal gas and reversible process assumption. Results shows less than 5% increase in the exergetic value of the turbine-burner to the conventional turbine. The energetic and entropic nature of the engine was studied, and a representation for deriving a quantifiable value meeting the criteria of the first and second laws was presented. It was shown that under isothermal conditions within the turbine-burner component the exergetic value can still be calculated. This is the case with an ideal gas assumption and obeying the second law rule stating that entropy can only be zero for a reversible process or positive for an irreversible process. An exergetic comparison between the conventional engine and the turbine-burner shows a significant increase for the turbine-burner. From the results of this study, the advantage of the turbine-burner compared to the conventional turbine is less than 5%.

❖ **Banakar, P. and Basawaraj[4]**

The author has presented computational analysis of flow in after burner diffuser mixer having different shapes of struts, analysis has been carried out using 45-degree sector model of the diffuser mixer without strut and with struts considering the periodicity of geometry. An unstructured grid has been generated and Simulation has been done using ANSYS FLUENT software, flow has been simulated by solving governing equation of mass, momentum, energy. Turbulence closure is achieved with k-epsilon turbulence model with standard wall functions. The analysis has been carried out with velocity components, total pressure and total temperature at inlet boundary conditions and a mass flow rate at the outlet. The present study shows that presence of combination of both aerofoil and cylindrical struts gives better increase in pressure in afterburner's combustion section and also helps in guiding the flow. Increase of pressure in the flow with struts is beneficial as it leads to better mixing of air and fuel in the after burner unit and this gives efficient pressure gain for proper combustion. Aerofoil and cylindrical struts also acts like guide vanes for the flow and also converts swirl flow into stream line.

❖ **Bheemaraddi, S. B. and Kumarappa,s[5].**

The author has described assessment of turbulent boundary layer modeling methods by using computational fluid dynamics for gas turbine engine afterburner diffuser simulating the high temperature subsonic compressible flow in the diffuser duct of an afterburner unit of an air breathing jet engine. The diffuser duct was designed as per the guidelines explained in the standard engine design data hand book and corresponding experimentally measured results are referred for validation purpose. Aerofoil struts were incorporated in the design and the pressure loss due to the struts is assessed by CFD analysis. The CFD analysis is carried out with SST K-Omega Turbulence model and direct resolution of shear stress approach employed in Mentors Shear Stress Transport Turbulence model. The pressure loss computed for diffuser with various struts configurations and compared without strut. The SST k-omega model is found to be a

satisfactory physical model to give good predictions for various flow characteristics of an annular diffuser. Total Pressure loss is well predicted by the numerical model which has close agreement with experiment. The diffusion in the diffuser with struts is interrupted by the reduction of the flow area due to the struts and their wakes and this is constant after number of blades or struts are 12. The pressure loss is significantly increased by the presence of struts.

❖ **Haran, A. P., et al[7]**

The author has done a brief survey of analysis of an after burner in a jet engine. After burner is to provide a temporary increase in thrust, both for supersonic flight and for takeoff. A jet engine can produce more thrust by either accelerating the gas to a higher velocity or by having a greater mass (quantity) of gas. Afterburning has a significant influence upon engine cycle choice. Jet engines are usually run on fossil fuel propellant, and are thus a source of carbon dioxide in the atmosphere. Afterburners are generally used only when it is important to have as much thrust as possible. This includes takeoffs from short runways (as on an aircraft carrier) and air combat situations.

❖ **Haider, B. A., et al[8]**

The author has discussed Preliminary design of a short afterburner for single-spool expendable turbojet engine. The expendable turbojet engines are generally used in the reconnaissance air vehicles where the engine is positioned in the fuselage near the center of gravity. Consequently, a longer exhaust duct exists before of the nozzle section which can house a short afterburner. As a result of the limited length available for the afterburner, combustion chamber has to be considerably shorter than those usually used in the afterburners. Afterburner pressure loss and efficiency is calculated iteratively by using afterburner inlet conditions. It is found that by adding the designed short afterburner, about 20% increase in the thrust is achieved at the expense of 50% increase in specific fuel consumption. This gain can be very useful in the take-o_ with extra payload and maneuvering and dash in hostile conditions for any UAV. The real time performances of any combustion chamber can only be obtained by testing and hence it is recommended to bench test the proposed design. This testing has to be done over a complete operating envelope to ensure the consistency in the predicted performance. The CFD simulation of a 3D model can also give a good approximation of the output.

- **Isaac, J. J., et al[9]**

The author has presented a methodology for afterburner evaluation. This evaluated the afterburner configuration on the basis of flame stability, combustion efficiency and total pressure loss. An evaluation methodology, which was formulated, has been employed to arrive at design modifications for improved performance.

1. The gutter should be shifted away from the diffuser annulus to reduce the total pressure loss.
2. The gutter blockage could be reduced by removing two inner petals to reduce total pressure loss.
3. The gutter annular ring is not necessary for flame spreading and could be redesigned as a streamlined tube which acts as a communication passage for the ignited flame kernel to travel. Hence, the drag and corresponding pres-sure loss can be reduced.
4. End plates should be incorporated at the radial ends of the gutter to avoid impairment of flame stability.
5. The length/diameter ratio of the afterburner duct appears to be adequate.

- **Liu, F. and Sirignano, W. A[12].**

The author has described turbojet and turbofan engine performance increases through turbine burners. This concept is extended to include not only continuous burning in the turbine but also “discrete” interstate turbine burners as an intermediate option. A thermodynamic cycle analysis is performed to compare the relative performances of the conventional engine and the turbine-burner engine with different combustion options for both turbojet and turbofan configurations. Turbine-burner engines are shown to provide significantly higher specific thrust with no or only small increases in thrust specific fuel consumption compared to conventional engines. Turbine-burner engines also widen the operational range of flight Mach number and compressor pressure ratio. The performance gain of turbine-burner engines over conventional engines increases with compressor pressure ratio, fan bypass ratio, and flight Mach number. The turbine-burner engines are capable of and favor operations at high compressor pressure ratios. Although conventional engines may have an optimal compressor pressure ratio between 30–40 for supersonic flight. The turbine-burner engines benefit t more from efficient, large bypass fans than the conventional engines. The bypass pressure ratio can be optimized for a given mission.

- **Prasath, M. S., et al[14]**

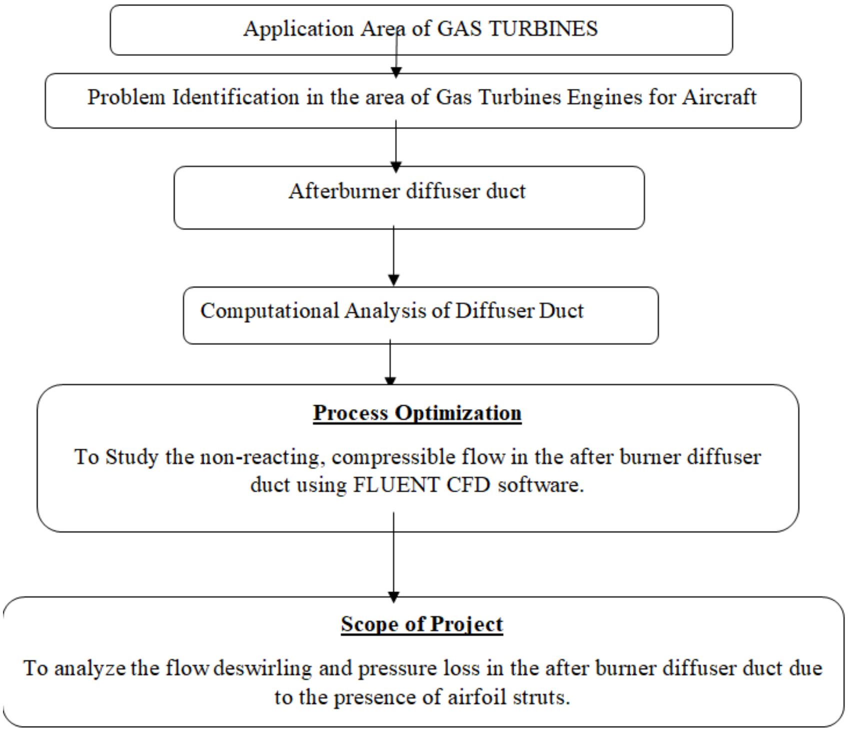
The author has focused CFD study of air intake diffuser. The air intake of the aircraft supplies the mass flow demand of the engine over a range of aircraft speeds and altitudes with high pressure recovery and minimum total pressure loss at the engine face. Also the duct must deliver air to the compressor 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. 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.

III. PROBLEM IDENTIFICATION

3.1 Performance Analysis

For any project it is necessary point that work to be done for increasing the Performance of device for achieving higher efficiency. In this work, the analysis have been done on the diffuser duct of afterburner of Gas Turbine Engine which 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.

This study helps in designing the diffuser duct of Afterburner for achieving higher performance



IV. METHODOLOGY

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.

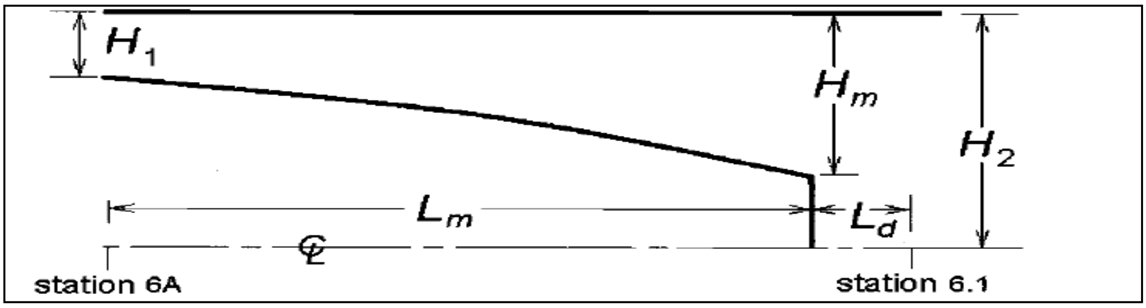


Fig 6: Geometry of the Diffuser duct (Reference, Aircraft Engine design, Jack D. Mattingly)

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**

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.

e-ISSN: 2582-5208
International Research Journal of Modernization in Engineering Technology and Science
(Peer-Reviewed, Open Access, Fully Refereed International Journal)
Volume:06/Issue:04/April-2024 Impact Factor- 7.868 www.irjmets.com

Phase.2

In this phase, the after burner diffuser duct is incorporated with eight number of airfoil struts. NACA 0012 symmetrical airfoil profile (Reference 04) is used in the design of struts.

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 deswirling due to the struts is also studied.

V. COMPUTATIONAL ANALYSIS

5.1 GEOMETRICAL MODELING AND GRID GENERATION:

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.

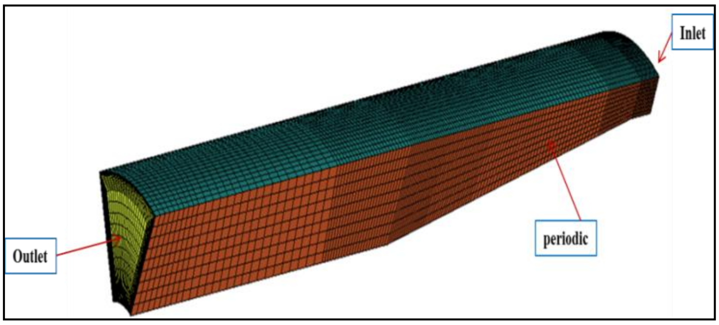


Fig 7: Computational domain of the after burner diffuser duct without struts(45 Degree Sector)

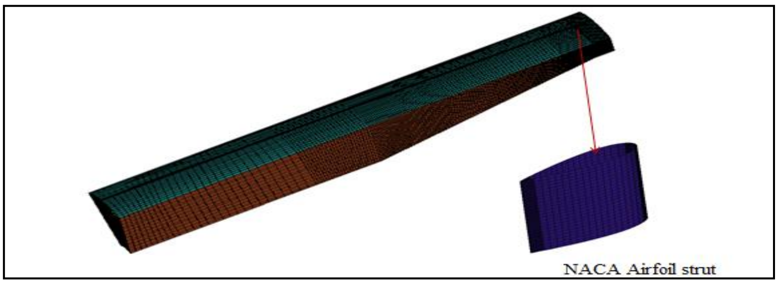


Fig 8: Computational domain of the after burner diffuser duct with struts(45 Degree Sector)

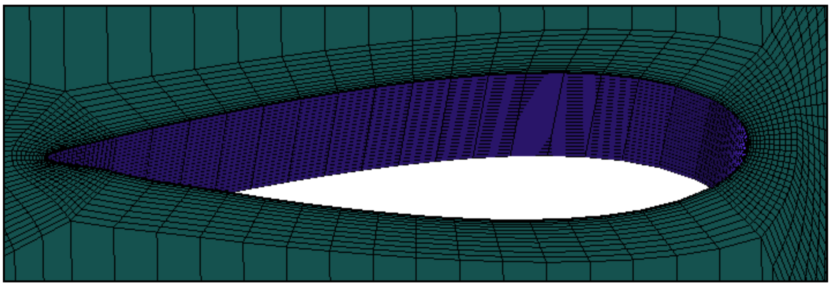


Fig 9. Body-fitted grid generated around the Airfoil Strut

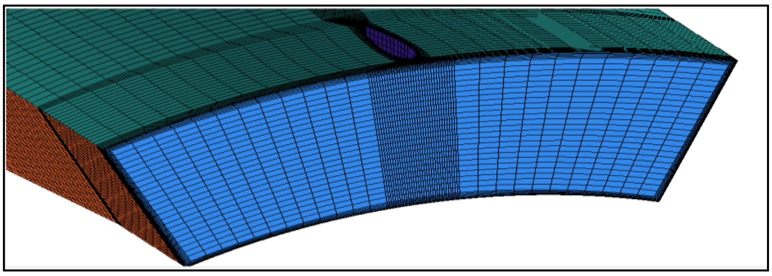


Fig 10: Mesh resolved near the wall region and around the airfoil to capture the viscous effects

e-ISSN: 2582-5208
International Research Journal of Modernization in Engineering Technology and Science
(Peer-Reviewed, Open Access, Fully Refereed International Journal)
Volume:06/Issue:04/April-2024 Impact Factor- 7.868 www.irjmets.com

The computational domain is discretised 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

5.2 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.

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

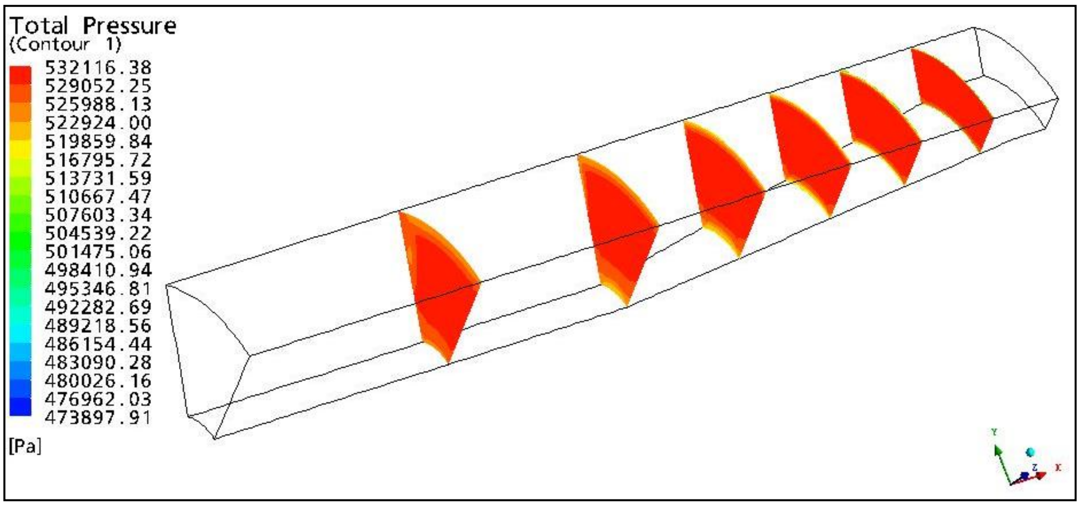


Fig 11: Contours of Total pressure (Without struts)

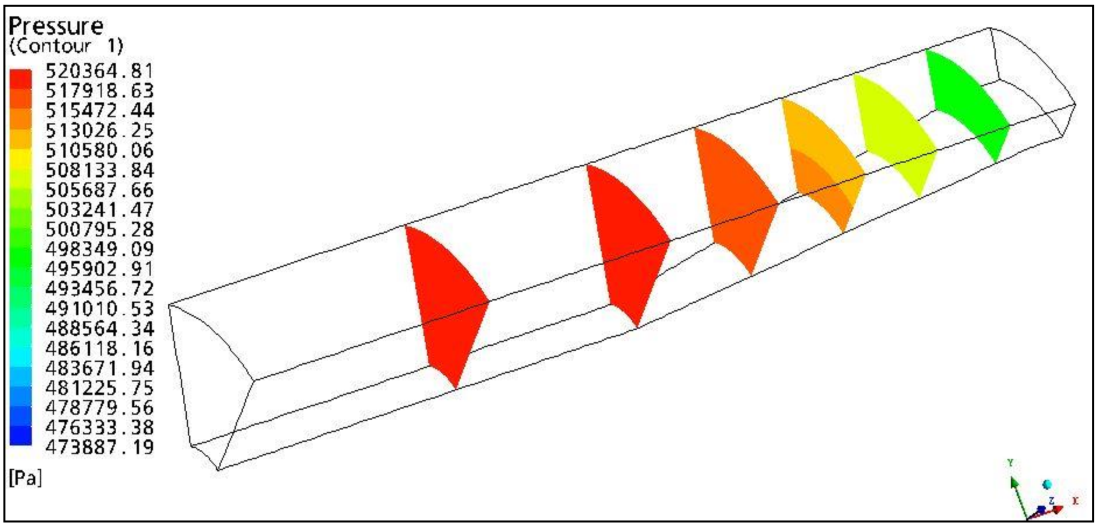


Fig 12: Contours of static pressure (Without struts)

e-ISSN: 2582-5208
International Research Journal of Modernization in Engineering Technology and Science
(Peer-Reviewed, Open Access, Fully Refereed International Journal)
Volume:06/Issue:04/April-2024 Impact Factor- 7.868 www.irjmets.com

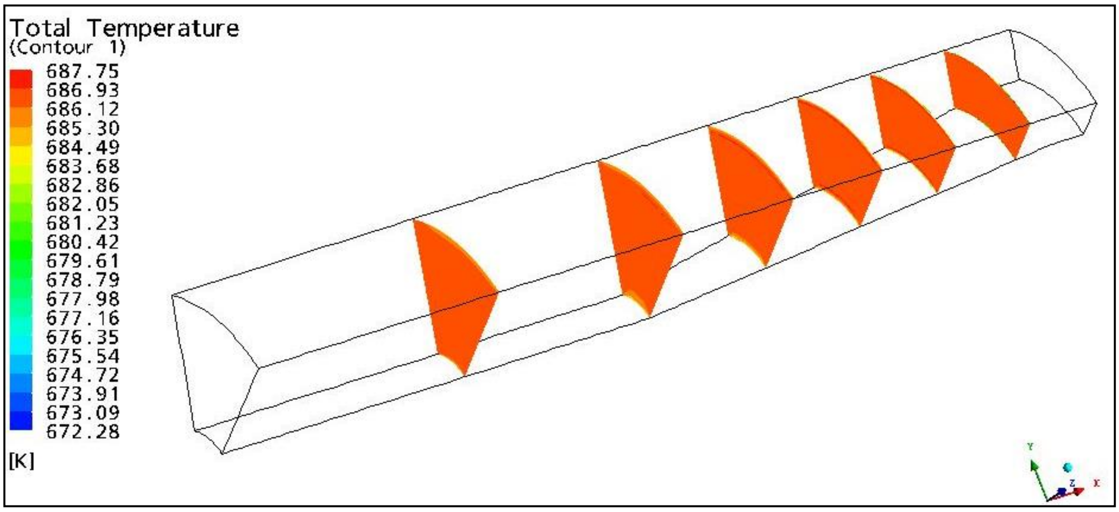


Fig 13: Contours of total temperature (Without struts)

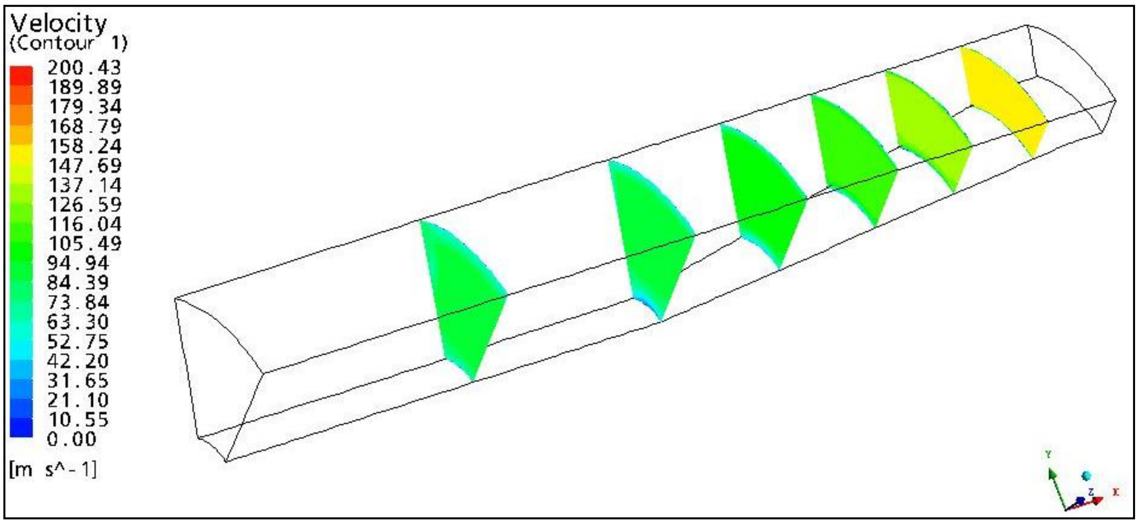


Fig 14: Contours of velocity (Without struts)

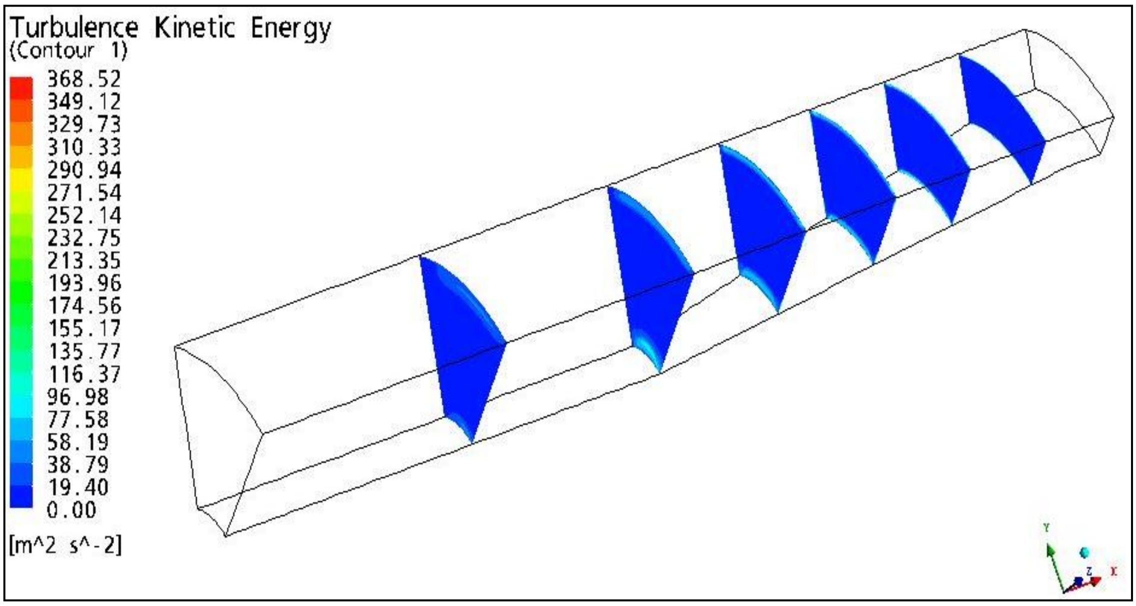


Fig 15: Contours of Turbulence Kinetic Energy (Without struts)

e-ISSN: 2582-5208
International Research Journal of Modernization in Engineering Technology and Science
(Peer-Reviewed, Open Access, Fully Refereed International Journal)
Volume:06/Issue:04/April-2024 Impact Factor- 7.868 www.irjmets.com

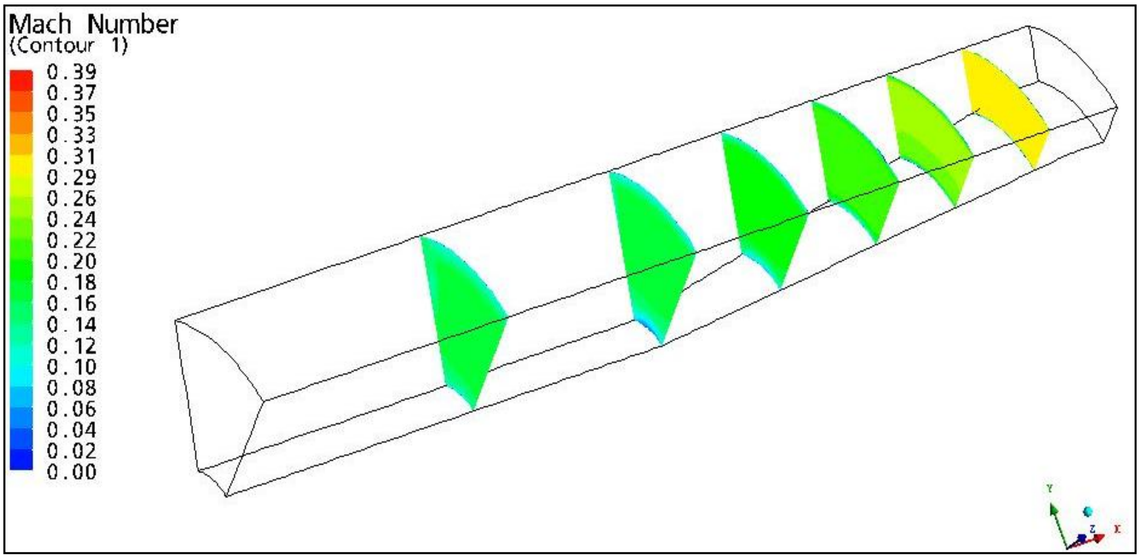


Fig 16: Contours of Mach number (Without struts)

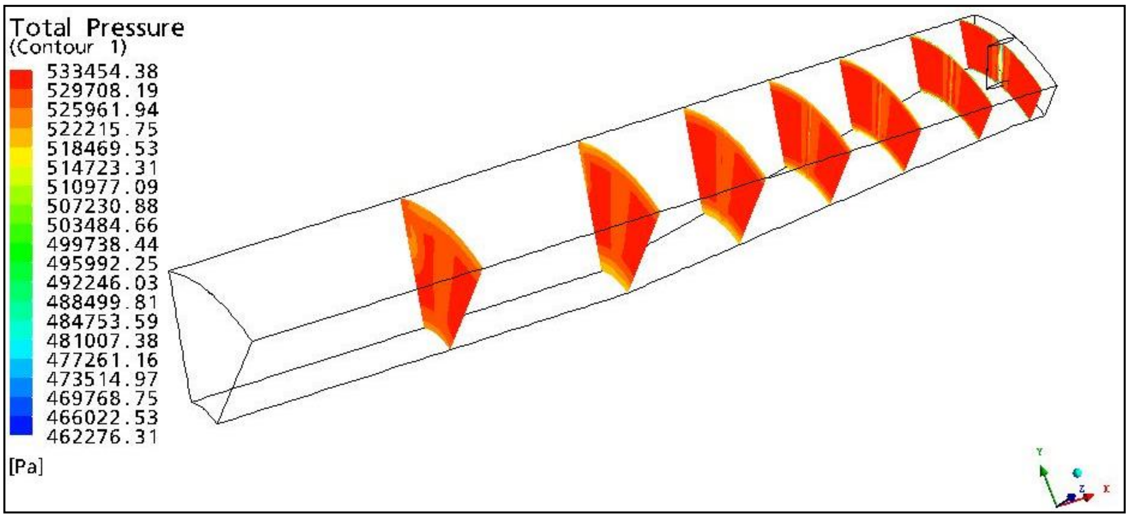


Fig 17: Contours of Total pressure (With struts)

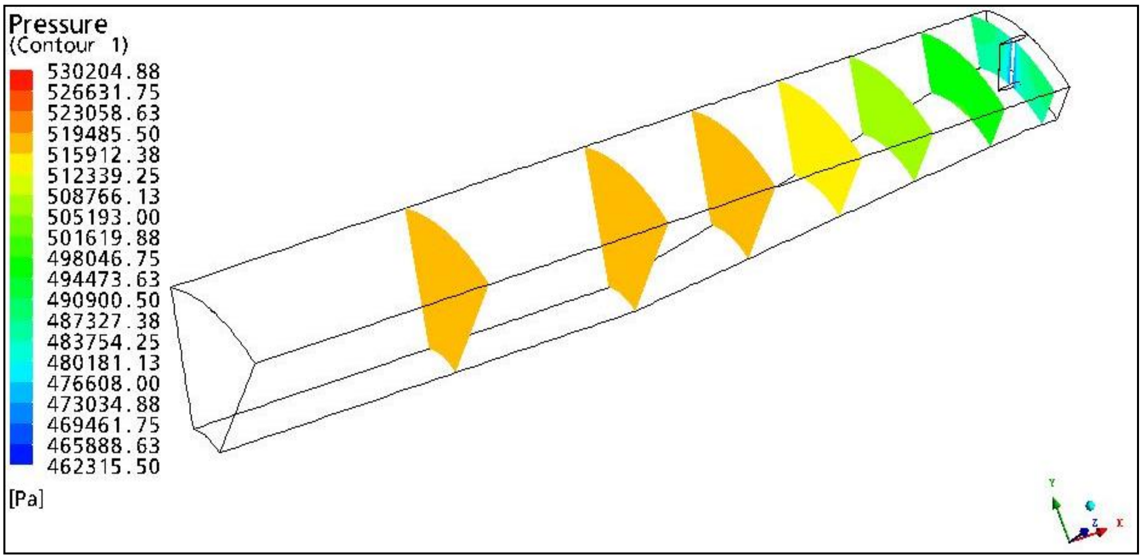


Fig 18: Contours of Static pressure (With struts)

e-ISSN: 2582-5208
International Research Journal of Modernization in Engineering Technology and Science
(Peer-Reviewed, Open Access, Fully Refereed International Journal)
Volume:06/Issue:04/April-2024 Impact Factor- 7.868 www.irjmets.com

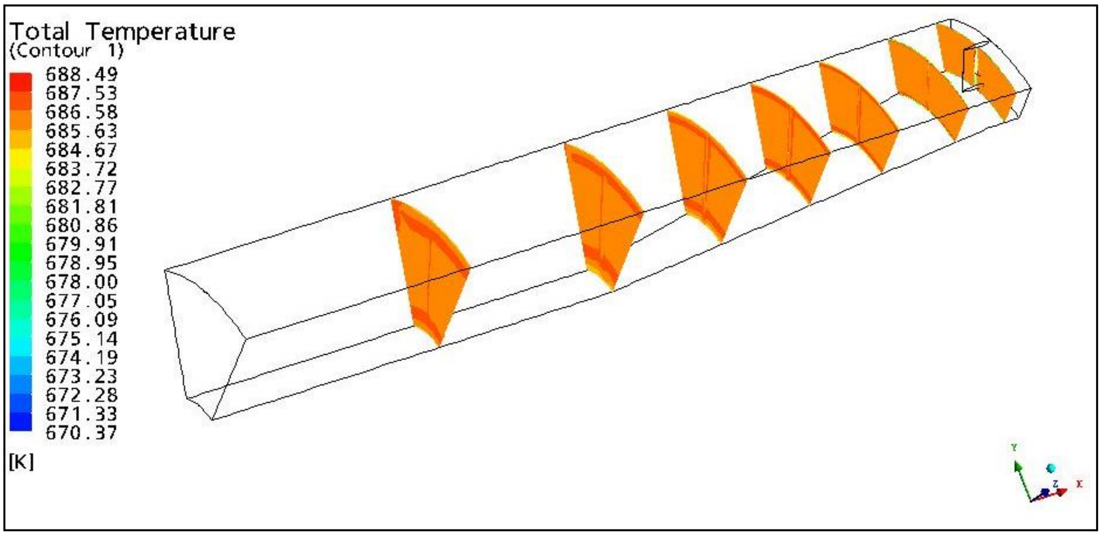


Fig 19: Contours of Total temperature (With struts)

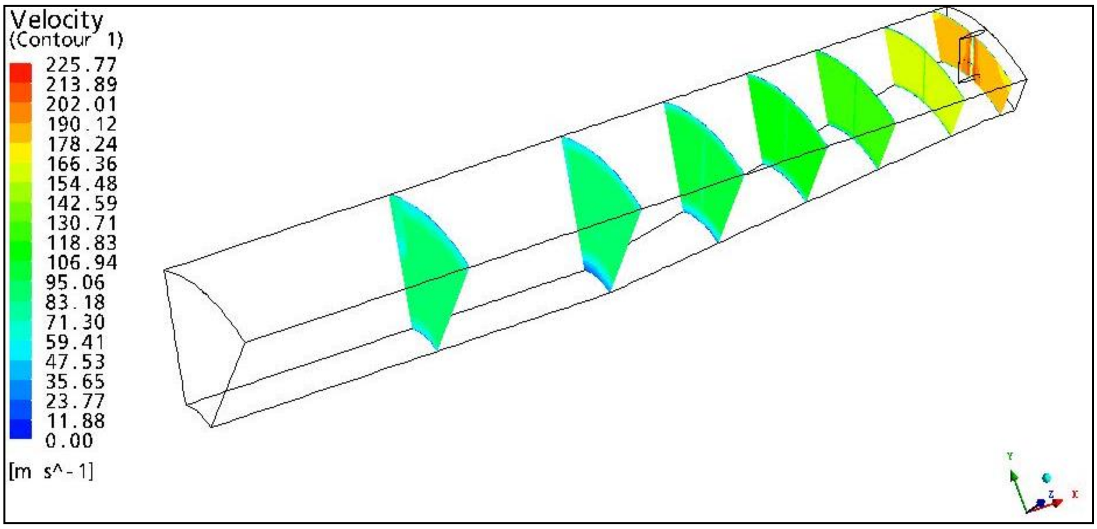


Fig 20: Contours of Velocity (With struts)

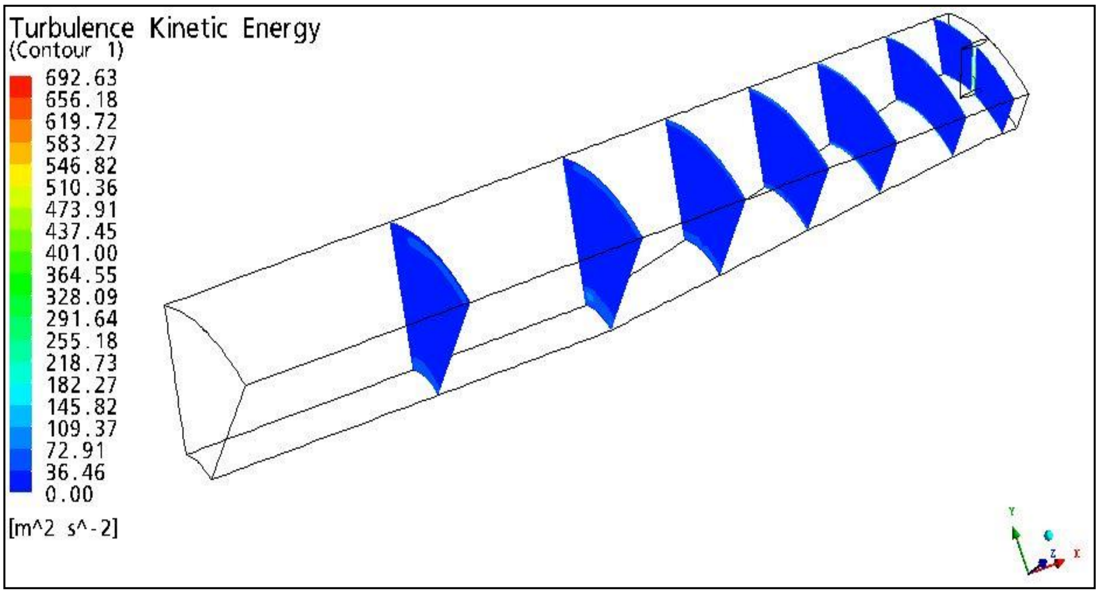


Fig 21: Contours of Turbulence Kinetic Energy (With struts)

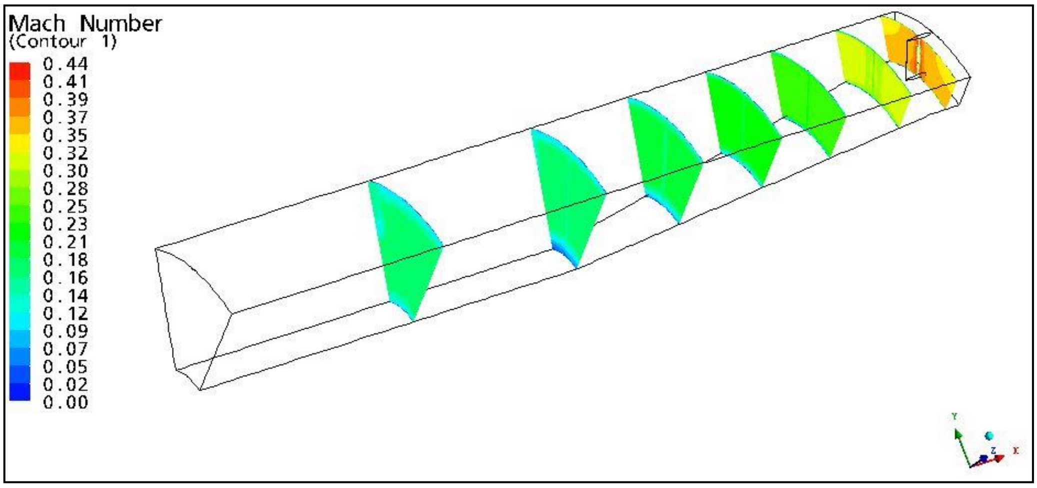


Fig 22: Contours of Mach number (With struts)

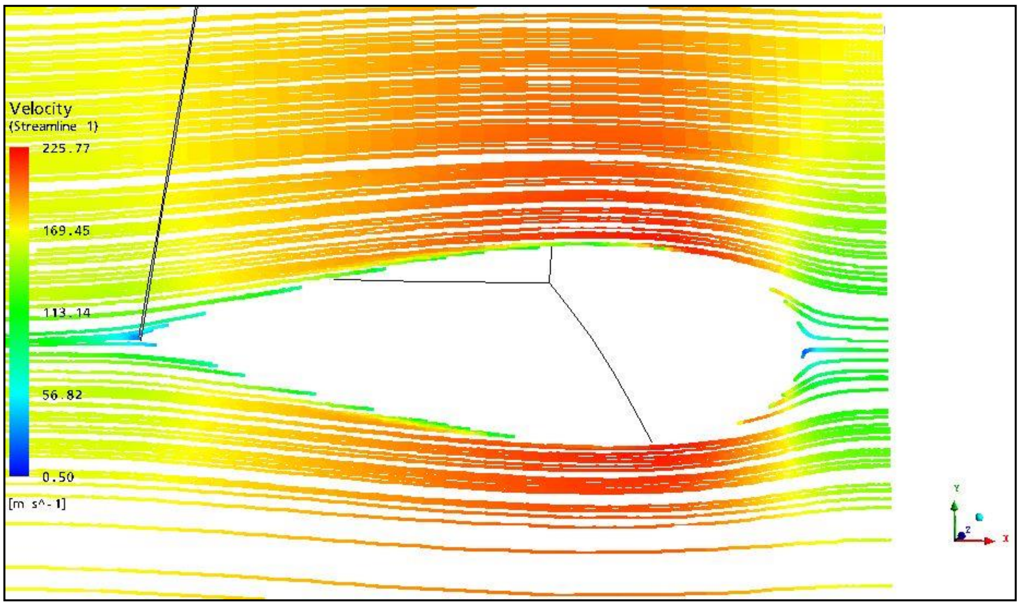


Fig 23: Stream lines around the airfoil strut

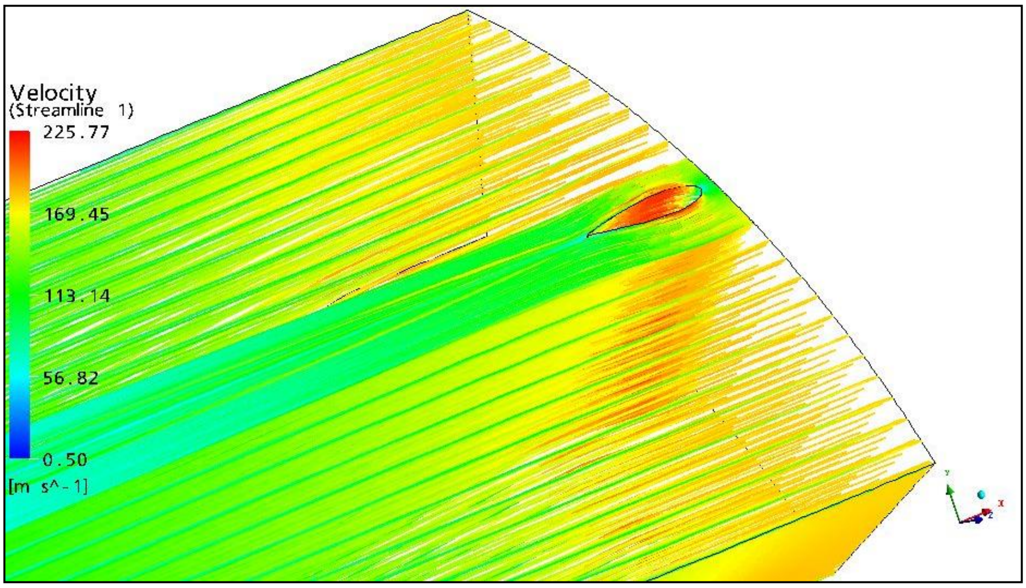


Fig 24: Stream lines around the airfoil strut (Sectional view)

e-ISSN: 2582-5208
International Research Journal of Modernization in Engineering Technology and Science
(Peer-Reviewed, Open Access, Fully Refereed International Journal)
Volume:06/Issue:04/April-2024 Impact Factor- 7.868 www.irjmets.com

VI. RESULTS AND DISSCUSION

6.1 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 result

Sl No	Grid size, Number of elements	Y-plus	Total pressure (absolute) at section 6.1,Pascals		Remarks
			Experimental	CFD	
Afterburner diffuser duct without struts					
1	80031	78	525249	503689	Y-plus value is high
2	135268	38		521963	Y-plus value is acceptable
3	157575	32		525003	Mesh finalized for CFD analysis
Afterburner diffuser duct with NACA 4412 airfoil struts					
4	219713	74	Not available	510697	Y-plus value is high
5	259871	35		521321	Y-plus value is acceptable
6	286961	34		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, 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.

6.2 CFD ANALYSIS RESULTS

Table 3: 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	Pascals	525249	525003
3	Absolute static pressure	Pascals	513920	512809
4	Total temperature	K	686.5	686.50
5	Mach Number		0.1817	0.1642

Table 4: Afterburner diffuser duct with strut

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

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 Pascal's.

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. This is shown in Fig 21 and Fig 22.

Fig 11 to 16 shows the flow field across various sections of the after burner diffuser duct without struts. Fig 17 to 24 shows the flow field across various sections of the after burner diffuser duct with NACA 0012 airfoil struts.

Fig 15 shows the contours of turbulent kinetic energy in the after burner diffuser duct without struts and Fig 21 shows the Turbulent kinetic energy contours with NACA 0012 airfoil struts. 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 6.1.

VII. CONCLUSION

1. CFD analysis is carried out for the after burner diffuser duct with and without struts

2. CFD analysis results for the after burner diffuser duct without struts is compared with the experimental data.

3. The contribution of NACA 0012 airfoil struts for pressure loss is estimated.

4. 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.

VIII. REFERENCES

[1] Abbott, "Theory Of Wing Sections "McGraw Hill, 1949

[2] Ameri, M., Dorcheh, F. J., "The CFD Modeling Of Heat Recovery Steam Generator Inlet Duct", International Journal of Energy Engineering (IJEE), Vol. 3, Iss.3, PP. 74-79, Jun. 2013

[3] Balogun, O., Changki, Mo., Mazher, A. K., "Exergy Analysis Of Gas Turbine Burner Engine", International Journal Of Scientific & Technology Research Vol. 3, Iss.1, Issn 2277-8616, January 2014

[4] Banakar, P., Dr. Basawaraj., "Computational Analysis Of Flow In After Burner Diffuser Mixer Having Different Shapes Of Struts", IJER, Vol.3, Iss.6, Issn2321-7758, Nov.-Dec. 2015

[5] Bheemaraddi, S. B., Dr. Kumarappa, s., "Assessment of Turbulent Boundary Layer Modeling Methods by Using Computational Fluid Dynamics for Gas Turbine Engine Afterburner Diffuser", International Journal of Innovative Research in Science, Engineering and Technology (An ISO 3297: 2007 Certified Organization), Vol.3, Iss.1, Issn.2319-8753, Jan. 2014

[6] Ebrahimi, H. B., "Overview of Gas Turbine Augmentor Design, operation And Combustion Oscillation", ILASS Americas, 19th Annual Conference on Liquid Atomization and Spray Systems, Toronto, Canada, May 2006

[7] Haran, A.P., Robert Lee, C. L., Raja, S. A., and Joseph, R.D., "Analysis of an After Burner in a Jet Engine", Advances in Aerospace Science and Applications, Vol.1, Number 1, PP. 7-22, 2011

[8] Haider, B.A., Davidovic, N., Durrani, N., "Preliminary Design of a Short Afterburner for Single-Spool Expendable Turbojet Engine", 49th AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition 4 – 7 Orlando, Florida, Aiaa 2011-575, Jan. 2011

[9] Isaac, J. J., Ramesh, N. R., Krishnakumar, V. S., Rajashekar, C., Shyamsundar, S. R., A.P. Haran, A. P., Sundararajan, V., "A Methodology for Afterburner Evaluation", 3rd National Conference on Air Breathing Engines and Aerospace Propulsion, IIT, Madras, 28-30th Dec. 1996.

[10] Islam, S. S., Rao, B. K. S., Chowdary, N., Naga, N., "Multidisciplinary Performance Analysis Optimization of Subsonic Flow on Turbine in a Diffuser Duct with New experimental Göttingen 535 Airfoil Strut and Present Usual NACA 4421Airfoil Strut", IJSR, Vol. 5, Iss. 3, Issn No. 2277-8179, March 2016

[11] Jack D. Mattingley "Aircraft Engine Design"AIAA Education series, 2001

[12] Liu, F., Sirignano, W.A., "Turbojet and Turbofan Engine Performance Increases Through Turbine Burners", Journal Of Propulsion And Power, Vol. 17, No. 3, May-June 2001

[13] Norris, T. R., "Test Program for High Efficiency Gas Turbine Exhaust Diffuser", Dec. 31 2009

[14] Prasath, M. S., Gowda, S. S., Senthilkumar, S., "CFD Study of Air Intake Diffuser", The International Journal Of Engineering And Science (IJES), Vol. 3, Iss. 01 PP. 53-59 , 2014, ISSN (e): 2319 – 1813 ISSN (p): 2319 – 1805

[15] Rajkumar, M., Ganesan, V., "Analysis Of Non Reacting Flow In An Aircraft Gas Turbine Engine Afterburner Model Using Finite Volume Method", Indian Journal Of Engineering & Materials Sciences, Vol. 10, PP. 341-352, Oct. 2003

[16] Selley, L., Poinsotz, G. L. T., Kaufmann, P., Krebs, W., Veynantek, D., "Large-eddy simulation of turbulent combustion for gas turbines with reduced chemistry", Center for Turbulence Research Proceedings of the Summer Program 2002

[17] Sheriff, N. M., "CFD Analysis Of Flow In Afterburner", Proceedings Of The 6th WSEAS International Conference on HEAT and MASS TRANSFER (HMT'09)

[18] Unaune, S. V., Ganesan, v., "Analysis Of Reacting Flows In An Aero Engine Afterburner Using Computational Fluid Dynamics", Indian Journal Of Engineering & Materials Sciences, Vol. 11, PP. 31-37, Feb. 2004

[19] Yahya, SM., "Fundamentals Of Compressible Flow" New Age International Publishers, 2000.

[20] Yogesh, TV., Ganesan,S.,Sankara, K. K., "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.