ANALYSIS OF GAS TURBINE ENGINE THRUST AUGMENTATION

Deen Dayal Chouhan[1], Dr. Dhananjay Yadav[2]

[1] *Research Scholar, Department of Mechanical Engineering, SOE, SSSUTMS, MP, INDIA*
[2] *Associate Professor, Department of Mechanical Engineering, SOE, SSSUTMS, MP, INDIA*

ABSTRACT

The purpose of this work is to analysis of Flow statistics, which control flow, are complexly compiled because every part of the speed is reflected in each intensity and continuity statistics. The most complex problem to be solved is the role played by pressure. Pressure is reflected in all pressure measurements but apparently there are no pressure calculations. Addition over non-linear values in convective terms of pressure calculations causes additional problems to find a solution for the mathematical set. Problems related to linear computational line inconsistencies and interactions between transport statistics are dealt with by a duplicate solution strategy such as the SIMPLE (Invisible Method of Statistically Connected Statistics) algorithm. In this method of multiplication where other scalar is combined with force calculations, the calculations are performed sequentially.

Keyword: - Catalytic Combustion, Fuel/Air Mixture, Porous Media Reactor, Pressure.

1.1 Introduction

The history of Swedish turbine industry started manufacturing their own counter-rotating radial steam turbines in has always been an industrial hub and was known to be one of the biggest manufacturers of cannons in the world. However, this era came to an end in 1911 when bankruptcy was filed by the Nordic Artillery plants. This is when, two years later, the two brothers bought the entire industrial area to establish STAL in Even though STAL and De Laval were engaged in manufacturing different kinds of turbines and the application areas also varied, they were still partly competitors. The two companies decided to merge towards the end of 1950's and the industry was brought together in Finspång [12]. With time the company expanded and started developing in the area of gas turbines. During its course, the company had multiple. The design of the SGT-800 is a great combination of robustness and high reliability which along with the features of high exhaust temperature combined with high efficiency makes it an optimal choice for many customers and operations such as energy companies, independent power producers, cogeneration and combined cycle installations and also oil and gas industry. Apart from this, the SGT-800 also has the best emissions performance in the 40-60[13].

Low calorific value gases mainly include blast furnace gas, coke oven gas, coal seam gas, landfill gas, biogas, and other combustible gases, which are regarded as "exhaust gas" and directly discharge to the environment. It is difficult to handle by conventional free flame combustion and can lead to environmental pollution and energy waste. The low calorific value gases are usually burned by preheating the mixture of air/fuel with an auxiliary device as assistant equipment. Porous media combustion is a promising method for handling low calorific value gases, which can recuperate heat from the combustion zone to the fresh mixture of fuel/air by solid conduction and radiation. It has been the subject of much attention in the past few decades due to its characteristics of higher power density, flame speeds, efficiency, and its lower pollutant emissions, compared with free flame combustion [14]. Much research has been done on the structures and materials of porous media to examine how they maintain good flame stabilization. Foe example, two layer burners with different porosity ceramic blocks was designed to extend the flame stability limits and reduce pollution, and foams of different materials. This indicated that the flame stability limit and the maximum flame temperature could be increased with the increasing diameter of packed beads, while the foam materials have little effect on the monoxide (CO) emission. experimentally studied the effect of the gap between alumina (Al2O3) pellets (preheating zone) and silicon carbide (SiC) foam (combustion zone) on the performance of methane/air combustion and reported that the limits of flame stability could be extended with a proper gap length. reported that flame stability limits and flame temperature in a two-layer porous burner can be controlled by the equivalent ratio of methane/air mixture. Wang et al. [15] studied the fuel-rich combustion of methane (CH_4) in a double-layer porous burner filled with alumina (Al_2O_3) beads of different diameters. An optimal pellet diameter of 7.5 mm at the downstream was obtained with the highest syngas energy conversion efficiency. Investigated the extra-lean filtration combustion of propane/air in porous media and showed that the average flame velocity increased and the temperature difference between the solid and gas becomes smaller [16].

Catalytic combustion of fuel/air mixture within a porous media reactor is an approach that can enhance combustion efficiency and decrease pollutant emissions, as the porous media acts as a special reaction place and supports the catalyst. There are few studies focused on combustion in catalytic and non-catalytic porous media [17]. Most of the previous research is focused on combustion in catalytic monolith type burners with different channel geometries developed the model of combustion of methane within both the catalytic and non-catalytic packed-bed reactors by employing single [17-step and multi- step reaction mechanisms for both gas-phase (homogeneous) and catalytic surface (heterogeneous) reactions. Shahamiri and Wierzba (2010) further studied the effect of the addition of hydrogen to the biogas/air mixture on combustion in a catalytic porous burner using detailed surface chemistry, which indicated that hydrogen can improve the oxidation of methane. studied the catalytic combustion of premixed methane/air in a two-zone perovskite-based alumina (Al_2O_3) pileup-pellets burner, and indicated that the flame stability limits increased with the increase of equivalence ratio or pellet diameter. [18] used a metal foam catalyst to catalyze methane (CH_4) combustion in a micro-combustion chamber and investigated the effects of inlet velocity and equivalence ratio on catalytic combustion characteristics of methane. These results show that a mixture of methane/air has an equivalent ratio of 1.0 and that a mixed fuel with an inlet velocity of 0.2–0.6 m/s can achieve combustion. Yang et al. (2020) numerically simulated the uniform combustion characteristics of methane/air in a semi-packed bed catalytic combustion chamber and found three combustion modes: completely heterogeneous combustion, heterogeneous combustion, and incomplete heterogeneous combustion [19].

2. Geometric Modeling And Grid Generation

In any flow problem it is important to define the visible boundaries that contain the liquid and the barriers to which the flow of liquid should occur. The Geometric model takes care of the above features by performing a calculation model. Different sources of geometric data are available to define the geometry of binding and barrier areas e.g. engineering drawings and information details created by computer-assisted design programs. The modeling effort depends on the complexity of the flow domain and often helps to simplify geometry where possible without sacrificing metaphorical accuracy.

Grid production is the next step in the process of mimicking CFD and involves subdividing the domain into a number of smaller, non-compliant domains commonly referred to as grid (or mesh) cells (or volume controls or elements.) Depending on the type of pricing strategy used. The number of cells in the grid controls the accuracy of the CFD solution. Grease grid, better accuracy. Both the accuracy and cost of computing depending on the computer hardware and calculation time depends on the quality of the grid. Accurate matches are usually not the same with fine meshing in sensitive areas of the domain where large gradients in the flow fluctuations are expected and coarser mesh in areas where the variance is very small. it can be of two types, formal or informal. In a well-organized grid the cells are well organized and a simple system (e.g., j, k indices) can be used to label features and identify.

neighbors. Organized grids come in several varieties depending on the shape of their cells. The simplest grid is produced by rectangular brick cells although their use islimited by the fact that geometric surfaces are usually approximated by blocking out entire cells, which leads to boundaries having discrete steps thereby introducing side effects. Better geometric representations of curved barrier areas can be achieved by modifying the grid elements to suit the stated geometric shapes, the emerging cells then have a normal hexahedral shape and the grid is often called a rectangular grid or by keeping rectangular cells. but they have added in some ways the descriptions of the obstacles that cross their path.

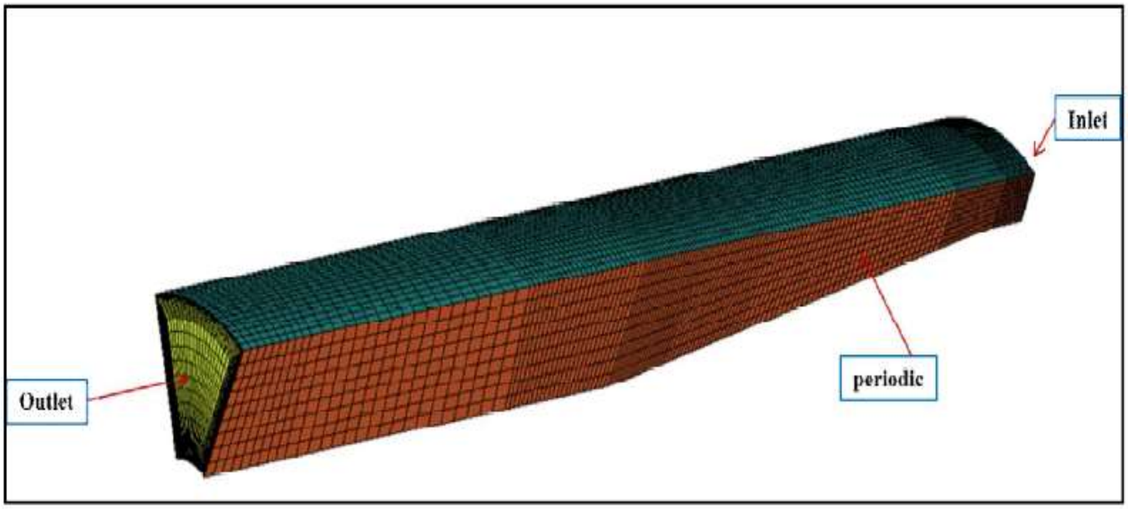

Fig 1: Meshing the model

In random grids the cells can be grouped in any way and special link lists must be kept to be visible to neighboring cells. Informal grids have a general advantage because they can be aligned with almost any desired geometry. However the grid production process is not completely automatic and may require significant user interaction to produce grids with acceptable levels of spatial adjustment while at the same time having a slight distortion of the feature.

Informal grids require more information to be stored and restored than systematic grids and changing the types of elements and sizes can increase pricing errors. The most popular type of unstructured grid contains tetrahedral elements. These grids are usually easier to make than those made of hexahedral elements, but usually have a poor accuracy.

In short, the best choice of grid system depends on a number of factors, namely, Ease in production, memory requirements, numerical accuracy, flexibility to complicate complex geometry and local variability for high or low resolution.

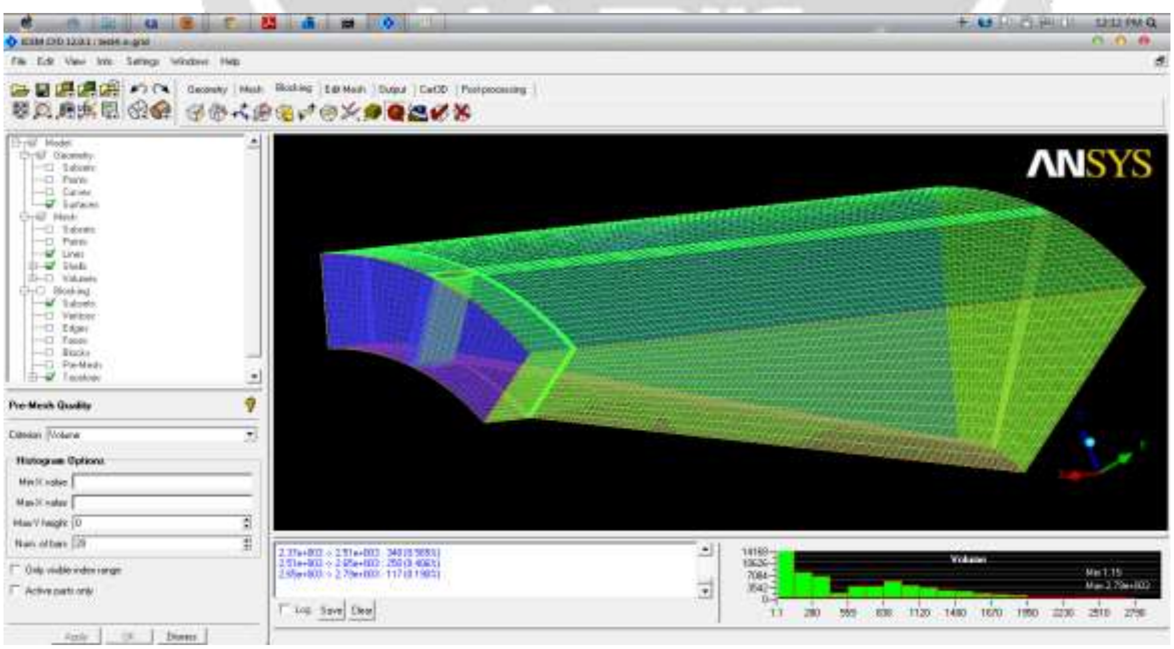

Fig 2: Meshing the Model in ANSYS

3. **Flow Specification**

Flow problem specification tells the CFD software the actual problem that needs to be solved and achieved by performing the following tasks:

Defining Liquid Properties: Liquids have a variety of properties and a solution must be provided for a specific method to calculate the values of this. Generally depending on the flow problem values are given as fixed or relationships with other independent variables are given or preferred local variations.

Determining the flow of the Calculated Calculations: The required variables depends on how the governing statistics are calculated and the algorithm set to be resolved. In addition, the flow environment and the type of models used to model the chaos, heat transfer etc determine the choice of flexibility.

Defining Border Conditions: Proper registration of boundary conditions and their modeling are the most important factors influencing the calculated outcomes. The type of boundary conditions required by any partial equation depends on the calculations themselves and how they have been classified. Some common boundary conditions, however, are met when CFD fluid flow problems are solved and are as follows.

Inlet: At the entry point, speed, pressure, bulk flow can be specified. Also, turbulence variables such as k and ε can be specified. Entry limits require a specific distribution of all floe variables.

Outlet: This marks a domain outlet. Normally, the gauge pressure is set to Zero where it exits. Parts of the speed and flexibility of the turbulence will have some zero point exit on the normal route to the exit boundary.

Symmetry: When the flow is relative to a particular plane there is no boundary flow and the output of some of the normal variables to the boundary is set to zero.

Wall: The most common boundary encountered in problems of clogged fluid flow. The shape of the slippery border is forced into the wall to flow viscous. The shear pressure and heat transfer between the liquid and the wall are calculated based on the flow data in the local flow field. In turbulent flow the adjacent wall area is usually modeled using semi-empirical formulas called “wall functions”. These functions close the viscosity-affected region between the wall and the flowing stream with full chaos. Bicycle or intermediate parameters: These parameters come in pairs and are used to determine if the flow has the same dynamic values in equal areas at both boundaries.

Explaining the Basic Conditions: Many solution algorithms require that the specific type of flow flow be specified in the solution. This may be because the flow depends on the time, at which point the initial variance is required for the calculation to start, or because of the use of a quasi-time variable algorithm solution. Equally, non-linear problem will require some initial guessing of the variance, which needs to be provided as a set of default values or user. When volatility is used it is usually set to a minimum positive or positive value.

4. Solution Procedure

There are a number of algorithms for solving various calculations, among them the SIMPLE algorithm for the correct correlation between pressure and speed and the line-by-line TDMA solution for algebraic arithmetic which are the most popular. The success of a numerical solution algorithm is determined by mathematical assumptions of integration and stability. In the following sections some words and strategies related to the numerical solution are discussed.

Convergence, consistency and stability

Integration: It is a feature of a numerical algorithm to produce a solution, which approaches a straightforward (analytical) solution if such a solution exists, as the volume control of the space or the size of the element is reduced to zero**.**

Consistency: The ability of a numerical system to generate algebraic mathematical systems, which can be shown to be equal to real control numbers, as the grid space is usually zero.

Stability: It is associated with a decrease in errors as the numerical method continues. The process is so stable that the calculations go to an integrated solution so that errors in the straightforward solution do not destroy the results with growth as the numerical process progresses.

Pressure-Velocity Coupling Solution Algorithm

From the first pressure field its main steps are as follows:

- Resolve the estimated pressure estimates to produce a central Velocity field.
- Solving the mathematical continuity in a mathematical way to correct the pressure.
- Pressure adjustment and Velocity.
- Solve some hidden scalar transport statistics.
- Repeat the above process until the merger is achieved.

Improvements to SIMPLE have produced more efficient and consistent duplication methods such as SIMPLER (SIMPLE-Revised) and SIMPLEC (SIMPLE-Consistent), the PISO algorithm, representing Pressure Implicit with

Splitting of Operators, contains an additional corrective action so that SIMPLE to improve its performance. With repetition. SIMPLEC and PISO have been shown to be as effective as SIMPLER in certain types of flow but it is not clear or categorized that better than SIMLER. Comparisons have shown that the performance of each algorithm depends on the flow conditions, the degree of integration and between the shutu and scalar calculations and the amount of slight relief used.

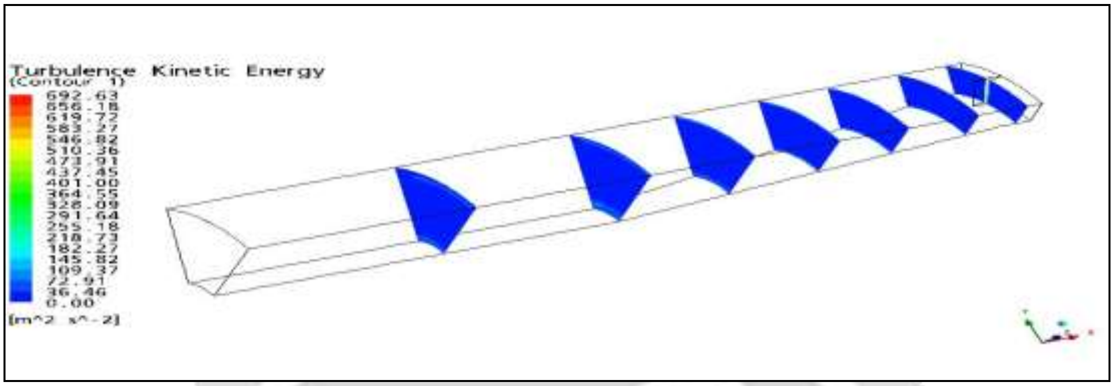

Fig 3.Contours of Turbulence Kinetic Energy (With struts)

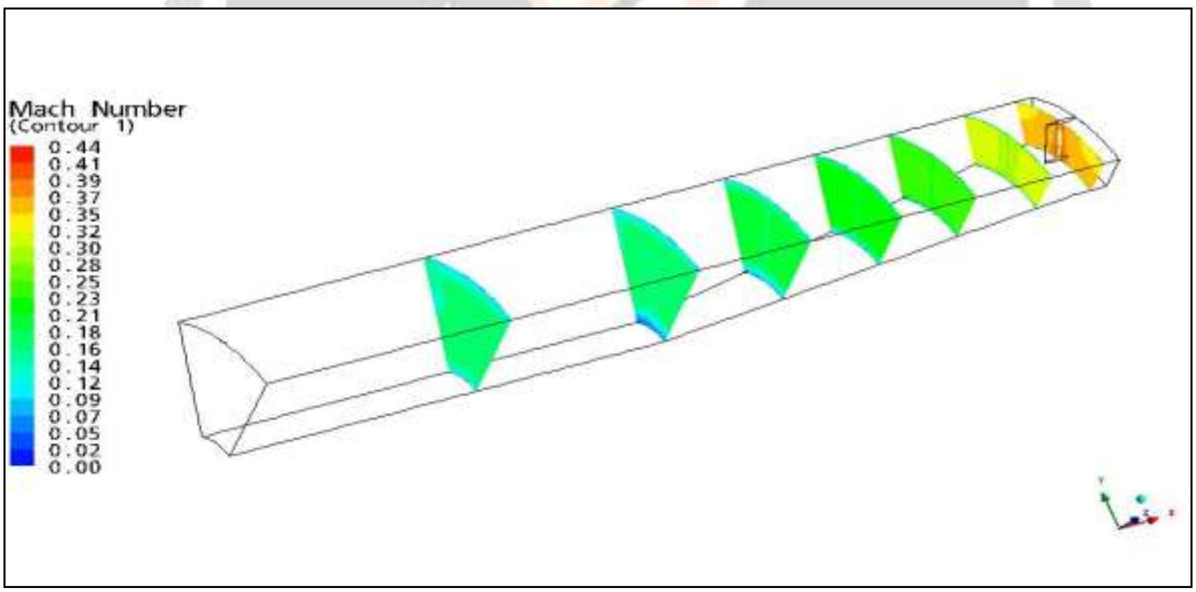

Fig 4.Contours of Mach number (With struts)

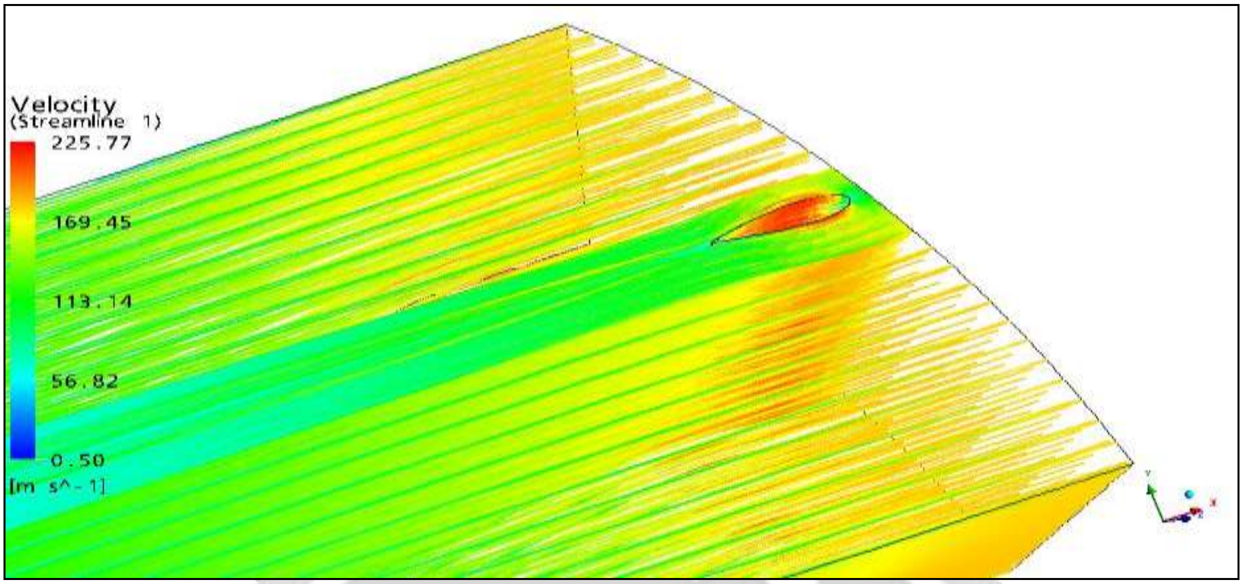

Fig 5.Stream lines around the airfoil strut

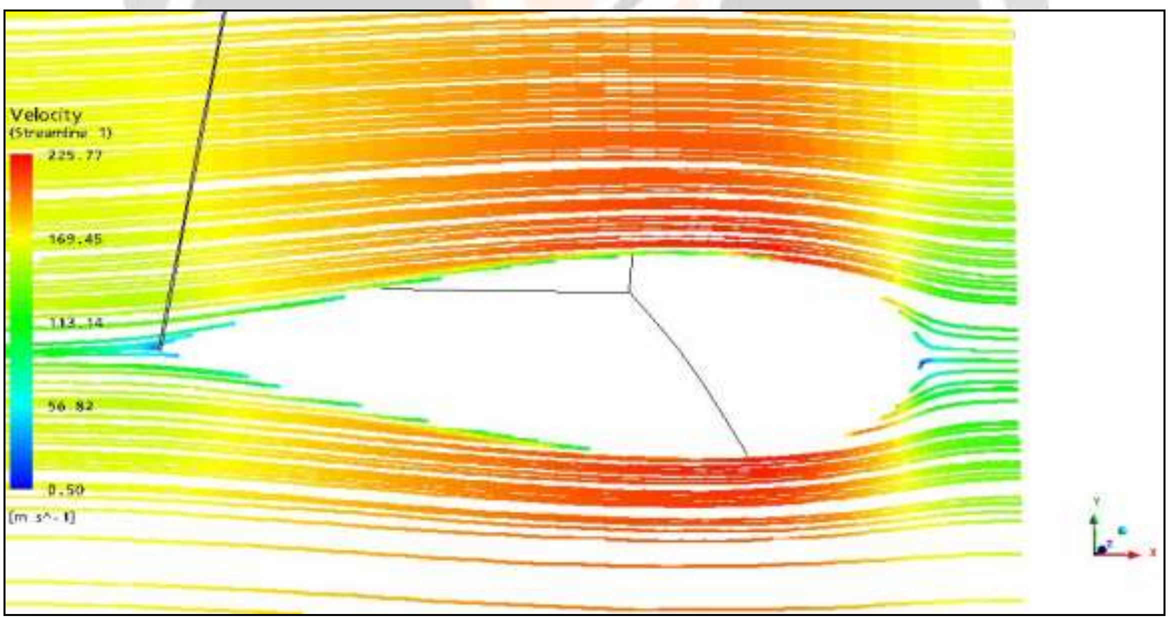

Fig 6.Stream lines around the airfoil strut (Sectional view)

Fig shows the flow field across various sections of the after burner diffuser duct without struts. It shows the flow field across various sections of the after burner diffuser duct with NACA 0012 airfoil struts.

shows the contours of Turbulent kinetic energy in the after burner diffuser duct without struts and 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.

5. Conclusion

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. The flame location of the catalytic porous burner was more sensitive to the flame velocity and insensitive to thermal conductivity compared to the inert porous burner. The distance of the flame location to the burner inlet is almost constant with the increasing length of the porous media for both the catalytic and inert porous burner, while the relative position of the flame location moved toward the upstream

6. References

[1.] K.M. Pandey, Member IACSIT and A.P. Singh, "CFD Analysis of Conical Nozzle for Mach 3 at Various Angles of Divergence with Fluent Software", IJCEA, Vol.1, No.2, August 2010.

[2.] Vinod kumar S Hiremath, S. Ganesan, R. Suresh, "Design And Analysis Of Exhaust Diffuser Of Gas Turbine Afterburner Using CFD", Proceedings of 27th IRF International Conference, 26th June, 2016. [10] Santhosh Kumar Gugulothu and Shalini Manchikatla, "Experimental and Performance Analysis of Single Nozzle Jet Pump with Various Mixing Tubes", International Journal of Recent advances in Mechanical Engineering (IJMECH) Vol.3, No.4, November 2014.

[3.] Karna S. Patel, "Flow Analysis and Optimization of Supersonic Rocket Engine Nozzle at Various Divergent Angle using Computational Fluid Dynamics", IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 11, Issue 6 Ver. IV (Nov- Dec. 2014), PP 01-10.

[4.] Kouichi Ishizaka, Susumu Wakazono (2003) "CFD Studies of Industrial Gas Turbine Exhaust Diffusers", Proceedings of the International Gas Turbine Congress Tokyo 2003.

[5.] SN Singh, V. Seshadri, K. Saha, K. K Vempati and S. Bharani (2006) "Effect of inlet swirl on the performance of annular diffusers having the same equivalent cone angle", Proc. Inst. Mech. Engineer, Part G, Journal of Aerospace Engineering Vol. 220, No.2, pp.129-143.

[6.] SJ Kline, DE Abbott, RW Fox (1959)"Optimum design of straight-walled diffusers", ASME Journal of Basic Engineering 91 321-330.

[7.] Bhaskharone, E.A (1991) "Finite-element analysis of turbulent flow in annular exhaust diffuser of gas turbine engines", ASME Transactions Journal of Fluid Engineering.

[8.] A.C. Benim, W. Zinser (1985)"Investigation into the finite element analysis of confined turbulent flows using a κε model of turbulence", Computer Methods in Applied Mechanics and Engineering, Volume 51, Issues 1–3, Pages 507–523.

[9.] Azad, R.S and Kassab, S.Z (1989) "Turbulent flow in conical diffuser: overview and implications", Physics fluids A 1 (3), 564–573.

[10.] V. Vassiliev, S. Irmisch, S. Florjancic (2002) "CFD Analysis of Industrial Gas Turbine Exhaust Diffusers", ASME Turbo Expo 2002 pp. 995-1013.

[11.] Okwuobi, P.A.C and Azad, R.W (1973) "Turbulence in a conical diffuser with fully developed flow at entry", Journal of Fluid Mechanics.

[12.] Hoffman, JA, Gonzalez.G (1984) "Effects of smallscale, high intensity inlet turbulence on flow in a twodimensional diffuser", ASME, Transactions, Journal of Fluids Engineering. Vol. 106, pp. 121-124.

[13.] Armfield, S.W and Fletcher, C.A.J (1989) "Comparison of k–e and algebraic Reynolds stress models for swirling diffuser flow", International journal of Numerical methods in fluids.

[14.] B. K. Sultanian, S. Nagao and T. Sakamoto (1999) "Experimental and Three-Dimensional CFD Investigation in a Gas Turbine Exhaust System", J. Eng. Gas Turbines Power Volume 121, Issue 2, 364.

[15.] Investigation in a Gas Turbine Exhaust system", J. Eng. Gas Turbines Power Volume 121, Issue 2, 364

[16.] N.V. Mahalakshmi, G. Krithiga, S. Sandhya, J. Vikraman and V. Ganesan (2007) "Experimental investigations of flow through conical diffusers with and without wake type velocity distortions at inlet", Experimental Thermal and Fluid Science Volume 32, Issue 1, Pages 133–157

[17.] S. Roga, K.M.Pandey, A.P.Singh, Computational Analysis of Supersonic Combustion Using Wedge-Shaped Strut Injector with Turbulent Non-Premixed Combustion.

[18.] R. Prakash, D. Christopher, K. Kumarrathinam, “CFD Analysis of flow through a conical exhaust diffuser”, International Journal of Research in Engineering and Technology, Volume 3, Issue 11, NCAMESHE-2014.

[19.] Parameshwar Banakar, Dr. Basawaraj, “Computational Analysis of flow in after burner diffuser mixer having different shapes of struts”, International Journal of Engineering Research, Vol. 3, Issue 6, 2015.

[20.] Venugopal M M, Somashekar V, “Design and Analysis of Annular Exhaust Diffuser for Jet Engines”, International Journal of Innovative Research in Science, Engineering and Technology, Vol. 4, Issue 7, July 2015.

[21.] Nikhil D. Deshpande, Suyash S. Vidwans, Pratik R. Mahale, Rutuja S. Joshi, K. R. Jagtap, “Theoretical & CFD Analysis Of De Laval Nozzle”, International Journal of Mechanical And Production Engineering, ISSN: 2320-2092, Volume- 2, Issue- 4, April-2014.

[22.] Ali Asgar S. Khokhar, Suhas S. Shirolkar, “Design And Analysis Of Undertray Diffuser For A Formula Style Race car”, International Journal of Research in Engineering and Technology, Volume: 04 Issue: 11 | Nov-2015.

[23.] Manoj Kumar Gopaliya, Piyush Jain, Sumit Kumar, Vibha Yadav, Sumit Singh, “Performance Improvement of Sshaped Diffuser Using Momentum Imparting Technique”, IOSR Journal of Mechanical and Civil Engineering, Volume 11, Issue 3 Ver. I (May- Jun. 2014), PP 23-31.

[24.] Masafumi Nakagawa, Atsushi Harada, “Analysis of Expansion Waves Appearing in the Outlets of Two-Phase Flow Nozzles”, International Refrigeration and Air Conditioning Conference at Purdue, July 14-17, 2008.

[25.] .M. Pandey, Member IACSIT and A.P. Singh, “CFD Analysis of Conical Nozzle for Mach 3 at Various Angles of Divergence with Fluent Software”, IJCEA, Vol.1, No.2, August 2010