STUDY OF GAS TURBINE ENGINE COMBUSTOR

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 study the measure of the pressure losses provided by airfoil struts using FLUENT CFD SOFTWARE. The simulation results indicated that the gas temperature in the inert porous medium is higher than that in a catalytic porous medium, while the solid temperature in an inert porous medium is lower than that in a catalytic porous medium. The flame moved toward the burner exit with the increasing diameter of the packed pellets at a lower equivalence ratio and moved toward upstream with the increased thermal conductivity of packed pellets.

Keyword: - Gas Turbine, Afterburner, Thrust Augmenter, and Exhaust Gas.

1.1 Introduction

The gas turbine afterburner is a thrust augmenter, which provides an increase in demand by re-heating the exhaust gas. The afterburner significantly raises the temperature of the exhaust gas to increase engine trust. The main combustor of a gas turbine engine heats only about 25 percent of the air. Thus, afterburners can heat up to 75 percent of the remaining air. Although afterburning is used for a short time, the afterburner is permanently installed and will transmit complete loss of pressure in the flow even when not in use (so-called dry state) and thus reduce concentration and increase engine (SFC) engine use. The afterburner contains an exhaust diffuser, fuel injector, V-gutter as flame stabilizer, chute liner, explosives and cooling holes and no vent. The aerodynamic features of the diffuser between the turbine outlet and the afterburner inlet have a significant impact on afterburner performance. This section, located at the bottom of the Low Pressure Turbine (LPT) outlet, has different purposes.

1.2 Gas turbine combustor

The hot gas from the turbine enters the rear heat exchanger. Gases are distributed in the diffuser to reduce speed. After slowing down the fuel is injected into the fuel injection rings and burned, the fire reinforcement is done with a set of radials and ring gutters. Flow rotation is required behind the fire extinguishers to obtain stable and complete combustion. Gases sent by convergence, nozzles are separated to increase speed after increasing the total temperature of the after burner. Large parts of the after burner which is a diffuser, fuel injection rings, flame retardant materials contribute to a significant blockage in the flow and cause a total loss of pressure during dry and wet operation. The gases coming out of the turbine are always circulating. As the swirl in flow causes an increase in total pressure and reduced afterburner performance. Gas extraction is important. Twisted struts applied to post-heat geometry are used to remove the flow. Therefore a detailed study of dimensional turbulent flow in the diffuser area, behind the fire extinguishers is required for the design and development and testing of after burner performance in jet engines.

2. Exhaust Gas Diffuser

It is mounted at the end of the compressor; expand the air to atmospheric pressure to avoid excessive debris. An important factor in improving the efficiency of the Turbine, it operates at very high temperatures. Therefore, the liquid can be extracted from a gas turbine unit much easier than without a separator. This is because in a nondiffuser system, atmospheric pressure often pushes the liquid back into the turbine, causing a backflow and causing a significant decrease in turbine performance. The diffuser helps in the exhaust of the exhaust gases, thereby reducing the work of the turbine that is needed to push the gases out. Therefore, the useful function of the turbine increases, thus increasing the efficiency of the turbo machinery system. Here the problem is find that the design (shape) of the exhaust diffuser plays a major role in increasing the efficiency of the turbine. Also the exhaust diffuser operates at high temperatures.

Therefore the heat pressure may be formed at a higher rate if the complete state of the diffuser is not considered. In this project, it focuses mainly on the conical exhaust diffuser which provides turbine efficiency. The design of the conical exhaust diffuser is developed by the UNIGRAPHICS CAD software which works much better than the remaining CAD software. Subsequent CFD analysis of conical exhaust diffuser was studied under high air flow velocity. ANSYS Fluent software is used for flexible fluid analysis of conical exhaust diffuser. After that, the thermal analysis of the conical exhaust diffuser was performed with Ansys software to measure the heat generated heat pressures within limits or not.

The use of gas turbines to generate electricity began in 1939. Today, gas engines are one of the most widely used energy-efficient technologies. Gas turbines are a type of internal combustion engine (IC) in which the combustion of combustible gas generates hot gases around the generator to generate energy. The production of hot gas during the combustion process, not the fuel itself makes gas engines a name. Gas engines can use a variety of fuels, including natural gas, petroleum and fossil fuels. Burning occurs continuously in gas engines, as opposed to recurring IC engines, where burns occur periodically B. Introduction About Computer Aided Design CAD is an important industrial technique widely used in many applications, including automotive, shipbuilding, and aerospace, industrial and construction . design, prosthetics, and much more. CAD is also widely used to produce computer-generated images of special effects in movies, advertising and technical manuals. The advent of modern technology and the power of computers means that even perfume bottles and shampoos are designed using techniques that were unheard of by the 1960s engineers. Because of its great economic value, CAD has become a major source of computational geometry research, computer graphics (both hardware and software), and a unique geometry. Current computer software packages range from 2D draft programs to solid 3D models and above.

Modern CAD packages can also allow three-dimensional rotation, allowing the view of the object to be designed at any angle you want, even internally looking outwards. Some CAD software is capable of creating a flexible mathematical model, in which it can be marketed as CADD. C. Choice of Failure Theory The results of well-documented experiments by different authors on different perspectives on failure, show that the distortion of energy theory predicts the delivery of greater accuracy. Compared to this high shear stress theory predicts the effects that remain on the safe side. Major primary stress theory provides conservative results only when the sign of the two main pressures is the same (2-D case). Therefore, the application of the main stress theory of pure torsion is excluded when the signal of the two main pressures is opposite. Examination of the fracture sample of the loaded sample until it breaks, shows that in gravity, failure occurs in 45-degree angle lines with the axis of the load. This indicates shaving failure. On the other hand, brittle objects, fractures in normal planes on the load axis, indicate that the normal stress level determines failure. In view of the above, it is internationally accepted that for solid objects, a high-level general stress theory is most appropriate. As for ductile materials, high shear pressure theory provides safer and easier to use results compared to reversible energy theory, so it is universally accepted as.

In today’s world fuel is very important factor for survival along with water. Fuel used for automobiles, cooking and power generation obtained from fossil fuels extracted from sea and earth. These fossil fuels are going to finish after few centuries. Hence it is very much important to find replacement for these fossil fuels with non-conventional fuel sources. Biogas generated from organic waste can be used as one of the alternative for fuel. Many researches are going on for effective production of biogas so that it can be applied as alternative fuel. Biogas is very much cheaper than conventional fossil fuels. This paper aims to design burner suitable for domestic cooking which will use biogas as a fuel.

To get maximum output from burner optimum design of burner is require. The optimum design aims to optimum dimensions, optimum number of holes on burner, proper mixture of air and fuel and most important the optimum fuel flow rate. For that Computational Fluid Dynamics is used to simulate combustion of biogas on burner and Genetic Algorithm is used to optimize design of burner. The numerical simulation results verified with other researches done by expert persons and data available in various sources. Development of such burners leads to more efficient use of biogas without its waste. This technology is very less costly which will be helpful for poor people of urban and rural part Returning residual flow to the outlet of the turbine, so as to properly feed the main part of the afterburner with almost non-rotating flow. Reduce the flow rate at the entrance to the afterburner fire chamber, to stabilize the main line flame. Direction flow to get a flow that fits well with the center line of the engine, which increases engine push.

The total geometry of the afterburner diffuser is basically determined by the desired flow of the Mach number up and down the flame stabilization phase. In the area of redesign design, this Mach number is chosen in the range of 0.2-0.3. The inlet-to-outlet location ratio of the diffuser is therefore straightforward on the basis of the continuity of the calculation. However the distribution angle and length required for this area measurement should be determined. In the afterburner exhaust diffuser, the outer wall of the diffuser also becomes the inner wall of the bypass duct, almost straight. Therefore the entire flow rate should be located on the inner side of the diffuser.

3. Afterburner System

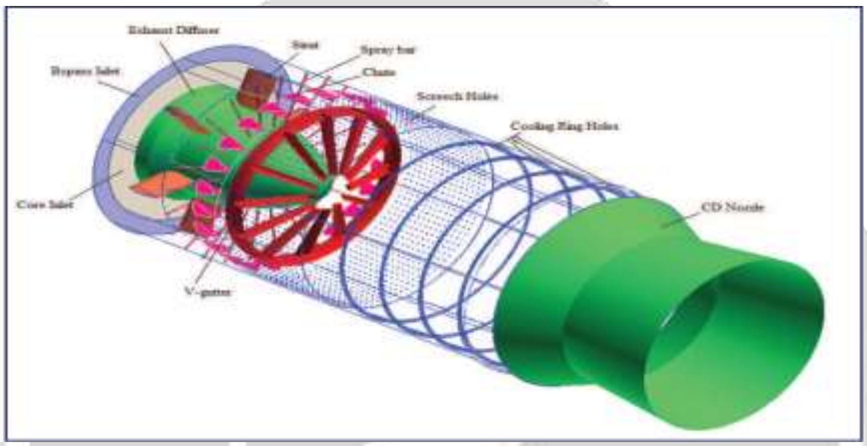

Fig 1.1.CAD model of the afterburner

The gas turbine afterburner is a thrust augmenter, which provides an increase in demand by re-heating the exhaust gas. The afterburner significantly raises the temperature of the exhaust gas to increase engine trust. The main combustor of a gas turbine engine heats only about 25 percent of the air. Thus, afterburners can heat up to 75 percent of the remaining air. Although afterburning is used for a short time, the afterburner is permanently installed and will transmit complete loss of pressure in the flow even when not in use (so-called dry state) and thus reduce concentration and increase engine (SFC) engine use. The afterburner consists of an exhaust diffuser, fuel injector, V-gutteras flame stabilizer, liner with chute, ripping holes and cooling holes and a nozzle. The aerodynamic features of the diffuser between the turbine outlet and the afterburner inlet have a significant impact on afterburner performance. This part, located at the bottom of the Low Pressure Turbine (LPT) outlet, has different purposes.

To restore the residual flow to the outlet of the turbine, to feed the main part of the afterburner is almost irreversible flow. Reduce the flow rate at the entrance to the afterburner heating chamber, in order to stabilize the main stream. Direction flow to get a flow that fits well with the center line of the engine, which increases engine push.

The total geometry of the afterburner diffuser is basically determined by the desired flow of the Mach number above the flame stabilization phase. In the heat design area, this Mach number is chosen at 0.2-0.3 diameter. The calculation of the inlet-to-outletarea ratio of the diffuser is therefore accurate on the basis of the continuity of the calculation. However the distribution angle and length required for this area measurement should be determined. In the afterburner exhaust diffuser, the outer wall of the diffuser also becomes the inner wall of the bypass duct, almost straight. Therefore all the flow rate should be located on the inner side of the diffuser.

4. Conclusion

The purpose of this work is to measure the pressure losses provided by airfoil struts using FLUENT CFD SOFTWARE. The simulation results indicated that the gas temperature in the inert porous medium is higher than that in a catalytic porous medium, while the solid temperature in an inert porous medium is lower than that in a catalytic porous medium. The flame moved toward the burner exit with the increasing diameter of the packed pellets at a lower equivalence ratio and moved toward upstream with the increased thermal conductivity of packed pellets. 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

5. References

[1.] 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)

[2.] 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.

[3.] Jack D.Mattingley et.al, "Aircraft Engine Design"AIAA Education series, 2001

[4.] Abbott, "Theory of Wing sections "McGraw Hill, 1949

[5.] SM.Yahya,"Fundamentals of Compressible flow "New Age International Publishers, 2000.

[6.] Dr.Isaac,J.J, Rajashekar.C ,N.R., Ramesh, et al, Afterburner flow visualization studies in a water tunnel, NCABE-paper-1992

[7.] Mattingly J.D, et al. Aircraft engine design, AIAA Educational series, and 1988.

[8.] Philip P. Walsh, et al .Gas Turbine Performance, 2nd edition 2004.

[9.] 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.

[10.] 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

[11.] Dr.N.Mohmed Sheriff, P.Selva Kumar, CFD Analysis of Flow in After Burner.

[12.] N Maheswara Reddy, E G Tulapurkara,VGanesan, Optimization of the Fuel Manifold Location inside a Jet Engine Afterburner, December, 2010.

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

[14.] 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.

[15.] 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.

[16.] 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.

[17.] 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.

[18.] 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.

[19.] 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.

[20.] 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.

[21.] 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