ijITECH International Journal of Innovative Technologies
WWW.IJITECH.ORG
ISSN 2321-8665
Volume.06, Issue.01,
January-June, 2018,
Pages:0327-0330

Design and Analysis of Conical Exhaust Diffuser

SEELI KANTHI REKHA[1], B. KISHORE KUMAR[2]
[1]PG Scholar, Dept of Mechanical Engineering Nalanda Institute of Engineering & Technology, AP, India,
E-mail: kranthirekhaseeli@gmail.com.
[2]Assistant Professor, Dept of Mechanical engineering Nalanda Institute of engineering & Technology, AP, India,
E-mail: kishorekumar@gmail.com.

Abstract: Exhaust Gas Diffuser is positioned at the end of the compressor; expand air to atmospheric pressure avoiding excessive residual thrust. Essential element to improve the Turbine efficiency, works at very high combustion temperatures. The fluid can, therefore be expelled from the gas turbine unit with more ease than in the absence of a diffuser. This is because in a system without a diffuser, the atmospheric pressure tends to push the fluid back into the turbine, causing backflow and resulting in a considerable drop in turbine performance. The diffuser assists in this expulsion of exhaust gases, thereby reducing the turbine work spent on pushing the gases out. Hence, the useful work of the turbine increases, thereby increasing the efficiency of the turbo machinery system. Here the problem find out that design(shape) of exhaust diffuser plays a major role of increasing turbine efficiency. Also exhaust diffuser works at high temperature conditions. So thermal stress may forms in huge amount if perfect shape of diffuser not considered. In this project, mainly focused on conical exhaust diffuser which gives more efficient of turbine. Design of conical exhaust diffuser is developed by UNIGRAPHICS CAD software which is more efficient than remaining CAD software. After that CFD analysis of conical exhaust diffuser studied under maximum air flow velocity. Ansys Fluent software is used for computerized fluid dynamic analysis of conical exhaust diffuser. After that, thermal analysis of conical exhaust diffuser was done through Ansys software for estimating produced thermal stresses are in within limits or not.

Keywords: CAD, Internal Combustion (IC).

I. INTRODUCTION

A. Gas Turbine

The use of gas turbines for generating electricity dates back to 1939. Today, gas turbines are one of the most widely-used power generating technologies. Gas turbines are a type of internal combustion (IC) engine in which burning of an air-fuel mixture produces hot gases that spin a turbine to produce power. It is the production of hot gas during fuel combustion, not the fuel itself that the gives gas turbines the name. Gas turbines can utilize a variety of fuels, including natural gas, fuel oils, and synthetic fuels. Combustion occurs continuously in gas turbines, as opposed to reciprocating IC engines, in which combustion occurs intermittently

B. Introduction About Computer Aided Design

CAD is an important industrial art extensively used in many applications, including automotive, shipbuilding, and aerospace industries, industrial and architectural design, prosthetics, and many more. CAD is also widely used to produce computer animation for special effects in movies, advertising and technical manuals. The modern ubiquity and power of computers means that even perfume bottles and shampoo dispensers are designed using techniques unheard of by engineers of the 1960s. Because of its enormous economic importance, CAD has been a major driving force for research in computational geometry, computer graphics (both hardware and software), and discrete differential geometry. Current computer-aided design software packages range from 2D vector-based drafting systems to 3D solid and surface modelers. Modern CAD packages can also frequently allow rotations in three dimensions, allowing viewing of a designed object from any desired angle, even from the inside looking out. Some CAD software is capable of dynamic mathematical modeling, in which case it may be marketed as CADD.

C. Choice of the Theories Of Failure

Well documented experimental results by various authors on the various theories of failure, indicate that the distortion energy theory predicts yielding with greatest accuracy. Compared to this maximum shear stress theory predicts results which are always on safer side. Maximum principal stress theory gives conservative results only if the sign of the two principal stresses is the same (2-D case). Therefore, the use of maximum principal stress theory for pure torsion is ruled out where the sign of the two principal stresses are opposite. When the fracture of a tension specimen loaded up to rupture is examined, it shows that for ductile materials, failure occurs along lines at angles 45 degrees with the load axis. This indicates a shear failure. Brittle materials on the other hand, rupture on planes normal to the load axis, indicating that maximum normal stress determines failure. Because of the above mentioned observations, it is universally accepted that for a brittle materials, the maximum normal stress theory is the most suitable. For ductile materials, the maximum shear stress theory gives conservative results and it is simpler to use as compared to distortion energy theory, so it is universally accepted as the

Copyright @ 2018 IJIT. All rights reserved.

theory of failure for ductile materials. But, where low weight is desired, the distortion energy theory is recommended.

II. LITERATURE REVIEW

R. Prakash [1] in his paper described that the exhaust diffuser of a fluid machine such as a gas turbine recovers static pressure by decelerating the flow and converting kinetic energy into pressure energy. It is hence a critical component in a turbo machine environment and plays a pivotal role in determining the performance of a turbo machine. Therefore, if the diffuser design is optimized for maximum pressure recovery, an increase in efficiency of the fluid machine can be brought about.

Parameshwar Banakar [2] in his paper stated that the subsonic flow analysis is carried out in diffuser mixer with struts and without struts. The total pressure loss, pressure gain, essential flow properties like Mach number, velocity, statics pressure, swirl are compared for both the cases. Analysis has been carried out using 45-degree sector model of the diffuser mixer without strut and with struts considering the periodicity of geometry.

Venugopal M M [3] in his paper discussed that Power and efficiency of the gas turbines depends highly on the performance of the exhaust turbine diffuser. To build a high efficient diffuser gas turbine we need to consider the flow through unsteady interaction with the high and low pressure rotating stage of the turbine, which create swirl flow. Study of various literature shows that there is room for improvement to enhance turbine performance. Since diffuser is the hub which deals with varying degrees of swirl. Swirl flow in diffuser section will create problems like, pressure loss during flowing across struts and reducing transitional Reynolds number renders the flow to turbulent which is not favourable. Reynolds number. Comparison of numerical results with experiments shows that the results of single- phase analysis is near to the experimental results.

III. 3D MODELING OF CONICAL EXHAUST DIFFUSER

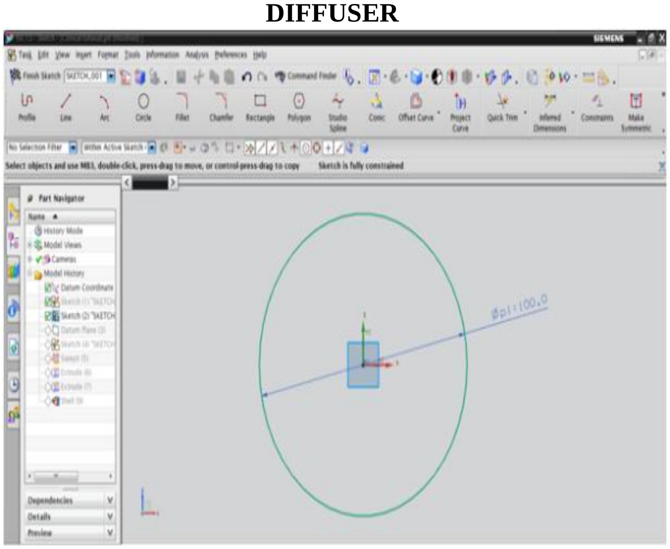

Fig.1.

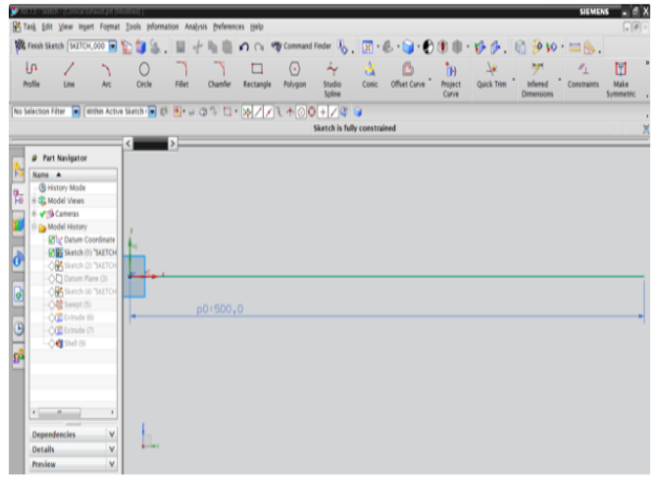

Fig.2.

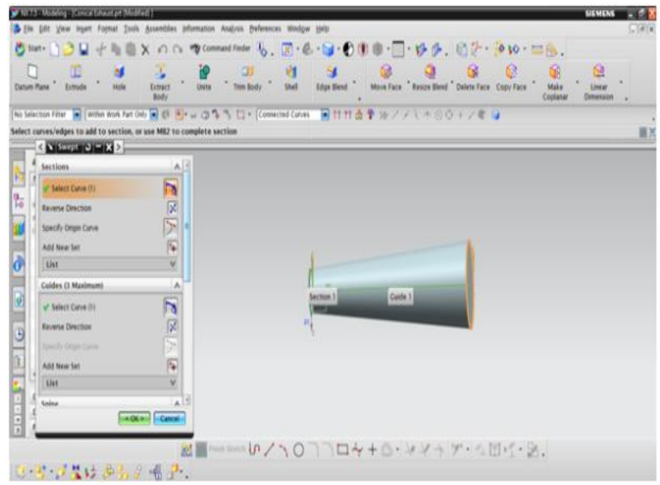

Fig.3. Shows swept option of conical exhaust diffuser.

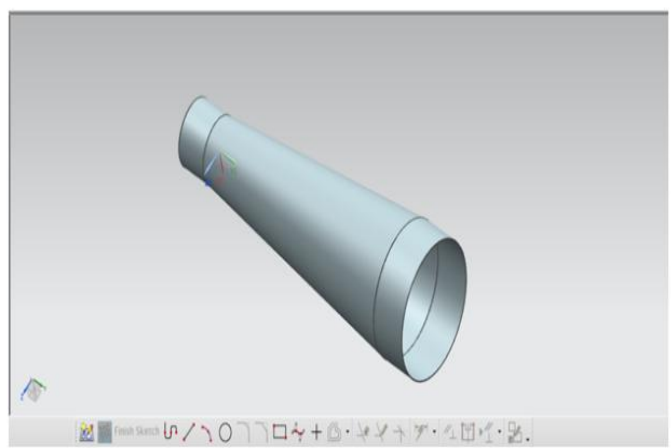

Fig.4.Shows Final model of conical exhaust diffuser.

IV. CFD ANALYSIS OF CONICAL EXHAUST DIFFUSERE

Bernoulli's principle is another key element of fluid dynamics, published in Daniel Bernoulli's 1738 book Hydrodynamica. Simply put, it relates the increase of speed in a liquid to a decrease in pressure or potential energy. For incompressible fluids, this can be described using what is known as Bernoulli's equation:

$$(v_2/2) + g_z + p/\rho = \text{constant} \quad \textbf{(1)}$$

Where g is the acceleration due to gravity, ρ is the pressure throughout the liquid, v is the fluid flow speed at a given point, z is the elevation at that point, and p is the pressure at that point. Because this is constant within a fluid, this means that these equations can relate any two points, 1 and 2, with the following equation:

$$(v_{12}/2) + g_{z1} + p_1/\rho = (v_{22}/2) + g_{z2} + p_2/\rho \quad \textbf{(2)}$$

The relationship between pressure and potential energy of a liquid based on elevation is also related through Pascal's Law.

A. CFD Analysis Of Conical Exhaust Diffuser

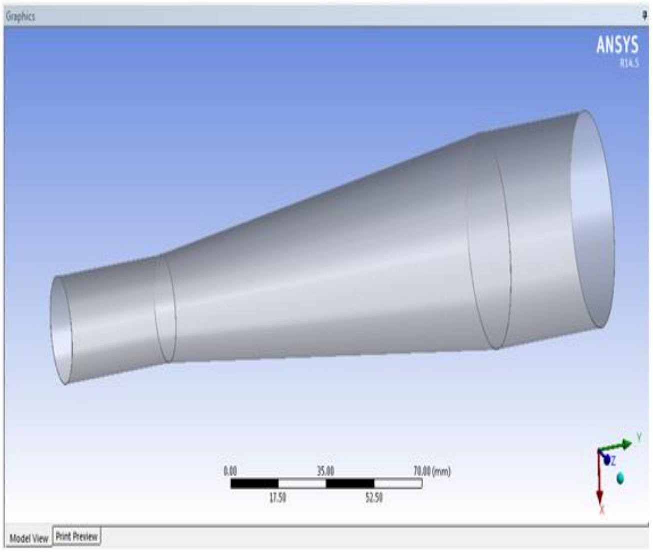

Fig.5.

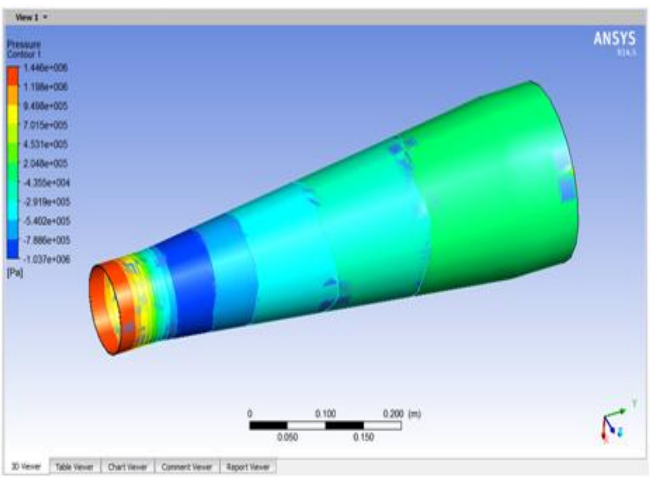

Fig.6.Pressure distribution.

V. THERMAL ANALYSIS OF CONICAL EXHAUST DIFFUSER

A thermal analysis calculates the temperature distribution and related thermal quantities in a system or component. Typical thermal quantities of interest are:

- The temperature distributions
- The amount of heat lost or gained
- Thermal gradients
- Thermal fluxes.

Thermal simulations play an important role in the design of many engineering applications, including internal combustion engines, turbines, heat exchangers, piping systems, and electronic components. In many cases, engineers follow a thermal analysis with a stress analysis to calculate thermal stresses (that is, stresses caused by thermal expansions or contractions).

A. Thermal Analysis Of Conical Exhausts Diffuser

Material properties of steel:
Density -7.80 g/cm3
Yield tensile strength -300MPa
Modulus of elasticity -200GPa
Poisson ratio - 0.3
Thermal conductivity -52W/m-K

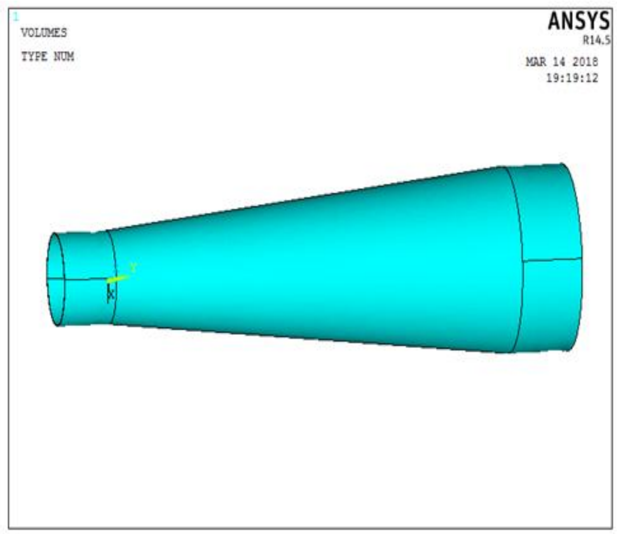

Fig.7.

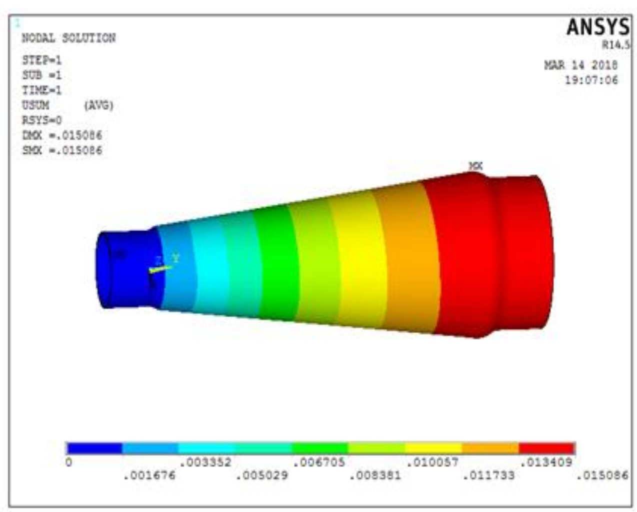

Fig.8.

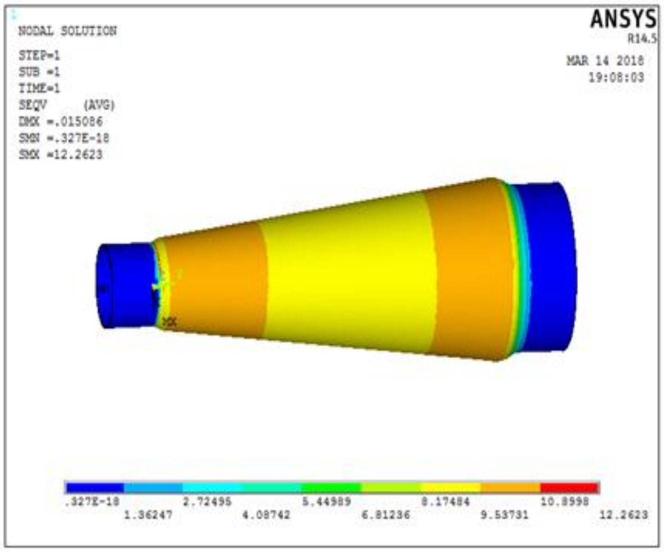

Fig.9.Thermal stress formed on exhaust diffuser pipe.

VI. RESULTS AND CONCLUSION

From CFD analysis results

Results type	Value
1. Maximum pressure distribution(MPa)	1.44
2. Maximum stream velocity in Gas turbine(m/s)	628

From thermal analysis in atmospheric pressure condition

Results type	Value
Maximum temperature distribution(K)	300
Maximum deformation (mm)	0.015
Maximum thermal stress(MPa)	12.26

VII. CONCLUSION

Pressure distribution foundin CFD analsys results is 1.44MPa and thermal stress found in exhaust diffuser is 12.26MPa which is lower than yield strength of steel (300MPa). So designed conical exhaust diffuser is safe under loading conditions.

VIII. REFERENCES

[1] 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.

[2] 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.

[3] 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.

[4] 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.

[5] 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

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

[7] 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.

[8] 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.

[9] 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.

[11] 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.