Scientific Journal of Impact Factor (SJIF): 4.72
e-ISSN (O): 2348-4470
p-ISSN (P): 2348-6406

International Journal of Advance Engineering and Research Development

Volume 4, Issue 6, June -2017

Modeling and Analysis of Conical Exhaust Diffusers using CFD

M. Santoshi Lakshmi [1], A. Vinutha[2]

[1] *Dept. Of Mechanical Engineering, Sanketika Institute of Technology and Management, Visakhapatnam*
[2] *Dept. Of Mechanical Engineering, Sanketika Institute of Technology and Management, Visakhapatnam*

Abstract — *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.*

The analysis is carried out for various models such as circular, square and hexagonal diffusers. The modeling is done in Creo Parametric 3.0, thermal analysis is done in Ansys 15.0 and Flow Analysis is carried out in Fluent 15.0(Workbench 15.0). The thermal analysis is performed to check for the better material which can be used to the Diffuser in order to increase the performance. Computational fluid dynamics (CFD) analysis was performed on diffusers With different shapes and based on the results obtained the geometry that yielded the maximum pressure recovery was identified.

Keywords- *Exhaust diffuser, Maximum pressure recovery, Performance.*

I. INTRODUCTION

The exhaust diffuser of the gas turbine is used to reduce the velocity of the working fluid discharged from the turbine and hence increase its pressure. This would result in increasing of the pressure ratio across the turbine. Also, since the diffuser increases the pressure of the working fluid, the pressure gradient at the diffuser exit reduces. 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.

Over the past decade, there has been a sustained interest in the analysis of exhaust diffusers owing to the effect it has on the overall efficiency of a fluid machine system and hence considerable work on has been done in this sphere. Both numerical and experimental investigation has been performed on diffusers to a great extent with the aim of improving the pressure recovery and consequently the efficiency of the turbo machine system. The use of computational fluid dynamic analysis is also rapidly gaining importance with regard to work in this sphere. The accuracy of the results obtained has increased considerably with the advent of computational methods. The purpose of this study was to improve the performance of exhaust diffuser in consideration of the distortion of the flow of the turbine exit, the diffuser performance in the big flow rate and to verify of CFD accuracy.

II. REVIEW OF LITERATURE

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.

Nikhil D. Deshpande[4] in his paper described that the de Laval nozzles are mechanical devices which are used to convert the thermal and pressure energy into useful kinetic energy. The values of temperature, pressure and velocity should be available at every section of the nozzle so as to design the nozzle shape, insulation and cooling arrangements.

III. MODELING OF AN EXHAUST DIFFUSER

The modeling of an exhaust diffuser is done in Creo Parametric 3.0 modeling software. The three cross-sections such as hexagonal, square and circular models are taken. The model of a circular exhaust diffuser is as shown in the Fig. 1.

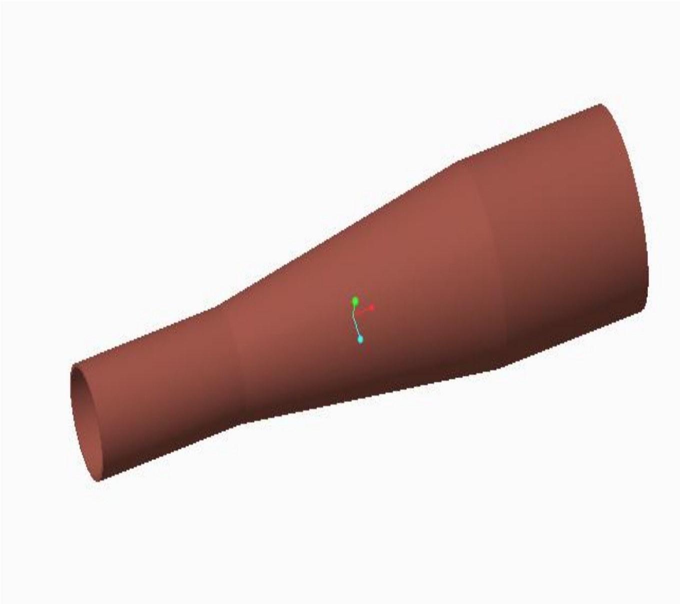


Fig. 1 Model of a circular exhaust diffuser

The drawing specifications of a circular exhaust diffuser are as shown in the Fig. 2.

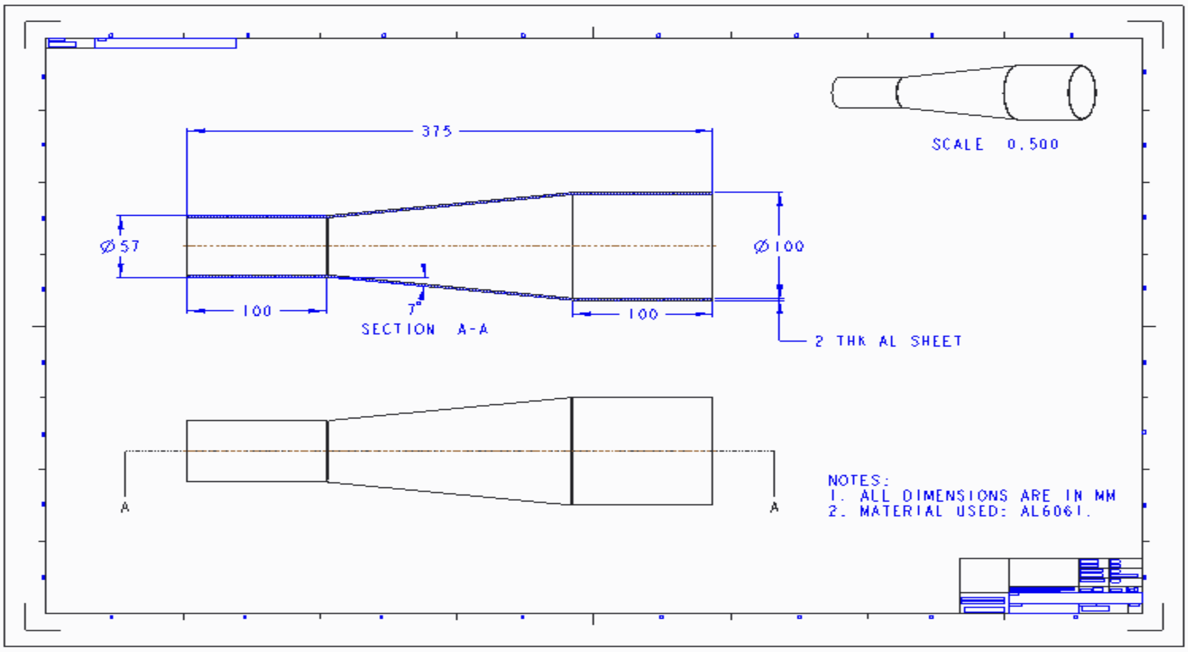


Fig. 1 Drawing Specifications of a circular exhaust diffuser

IV. COMPUTATION ANALYSIS

Computational Fluid Dynamics (CFD) is the science of predicting fluid flow, heat transfer, mass transfer, chemical reaction (e.g., combustion), and related phenomena by solving the mathematical equations that govern these processes using a numerical algorithm on a computer. The technique is very powerful and spans a wide range of industrial and non-industrial application areas.

All the CFD codes contain three main elements. They are as follows,

- Preprocessor.
- Solver.
- Post processor

The geometry is created in ANSYS ICEM CFD as per the given data for each of the model and a domain is created to encompass the flow inside the domain to the walls of the body. In order to study domain independence, three cylindrical domains are considered in trial and error method taking the distances from nose and tail ends of the model and taking the radius from the axis of the model. Three dimensional hexahedral grids were generated to discretize the body and the domain.

Three dimensional segregated implicit solvers is used in the present analysis, the k-ω, k-ϵ turbulence models in addition to the continuity and momentum equations were used as governing equations. Boundary conditions used in the present analysis are inlet as velocity inlet, outlet as Pressure Outlet, far field, and body as walls. All the three models are computed in the solver Fluent. The solution was stopped when changes in solution variables from one iteration to the next is negligible. Solution is iterated till the convergence is observed. Then forces and moments results were extracted from it. This data is saved as the data file in the solver itself.

Geometry and Domain are created in ANSYS 15.0. Blocking and Meshing is done. Checking the mesh quality and saving the file to solver Fluent. Export it into fluent software. Computing and monitoring the solution in Fluent. Examine and save the results. The geometric model for the circular exhaust diffuser is as shown in the Fig. 3.

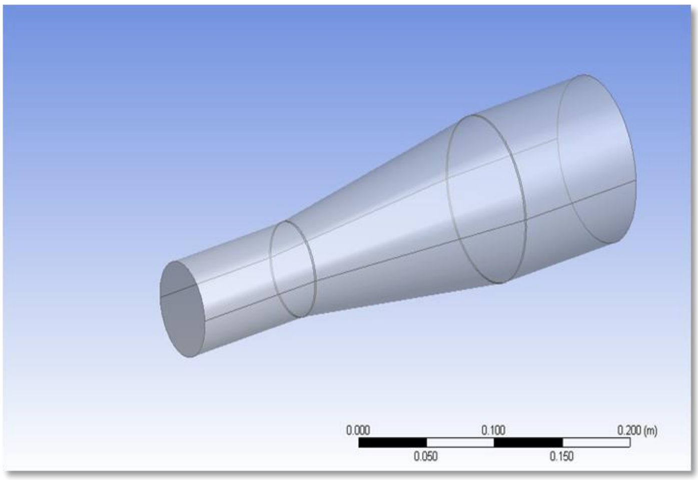


Fig. 3 Geometric model of the circular exhaust diffuser

The meshed model for the circular exhaust diffuser is as shown in the Fig. 4

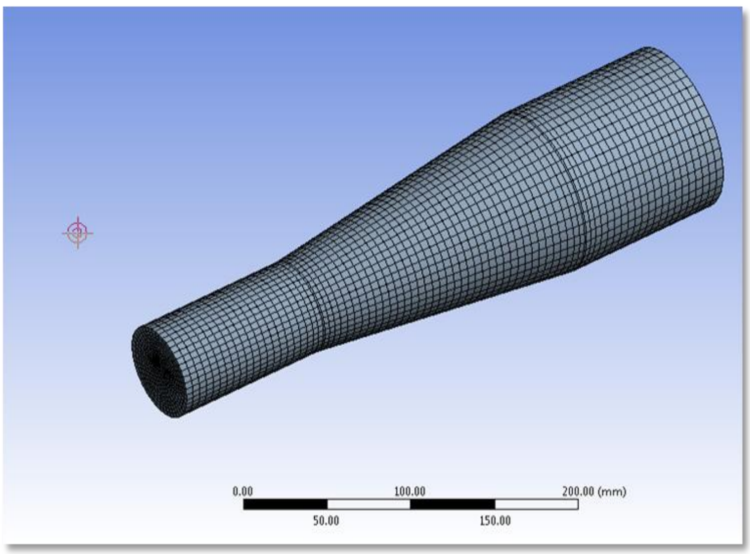


Fig. 4 Meshed model of the circular exhaust diffuser

The boundary condition for the exhaust diffuser is as shown in the Fig. 5.

S. No	ZONE	TYPE
1	Inlet	Velocity Inlet
2	Outlet	Pressure outlet
3	In_inner_wall	Wall
4	In_outer_wall	Wall
5	Boundary	Wall

Fig. 5 Boundary Conditions of the Exhaust diffuser

For the Inlet zone, the type would be velocity inlet. The Velocity inlet boundary conditions include the velocity of 45 m/s and a temperature of 1773 K. For the boundary, stationary wall conditions are taken. For the Outlet zone, the type would be pressure outlet. The pressure outlet boundary conditions are taken for standard temperature conditions and operating pressure conditions of 101325 Pa.

The Pressure Contours for the Exhaust diffusers are as shown in the Fig. 6

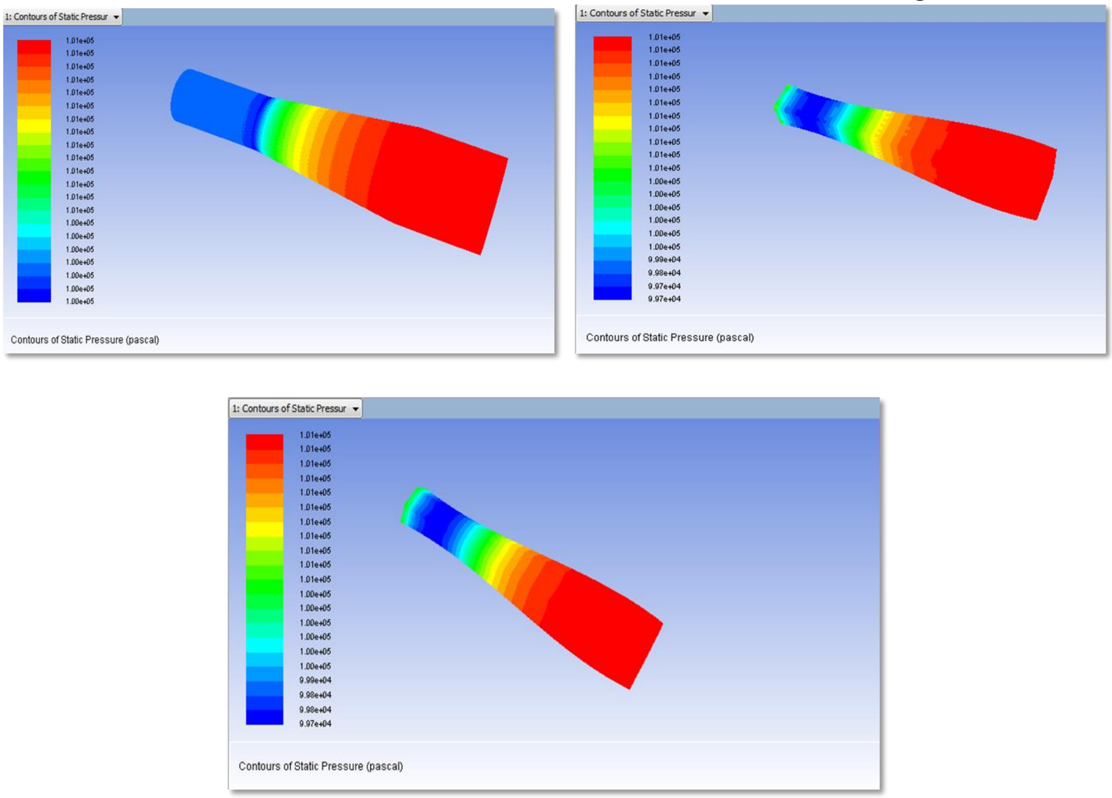
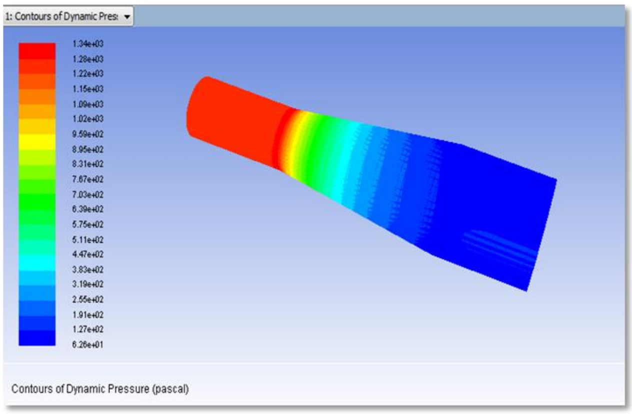


Fig. 6 Static Pressure Contours for the Exhaust Diffusers

The Dynamic Pressure Contours for the Exhaust Diffusers is as shown in the Fig. 7



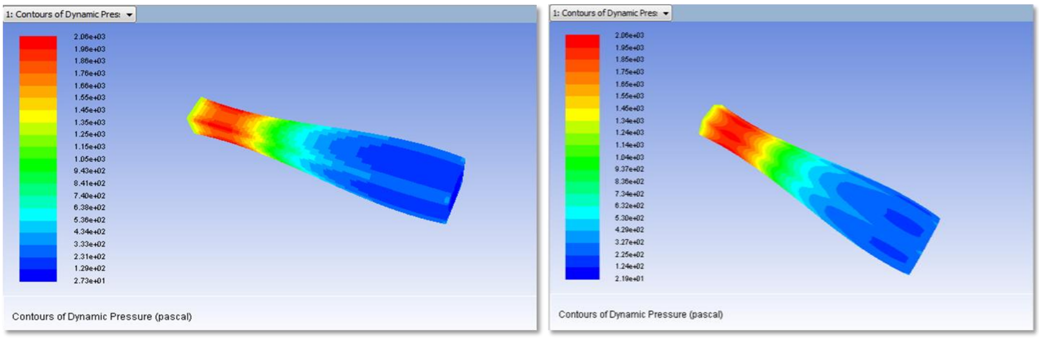


Fig. 7 Dynamic Pressure Contours for the Exhaust Diffusers

The Velocity Contours for the Exhaust diffusers are as shown in the Fig. 8

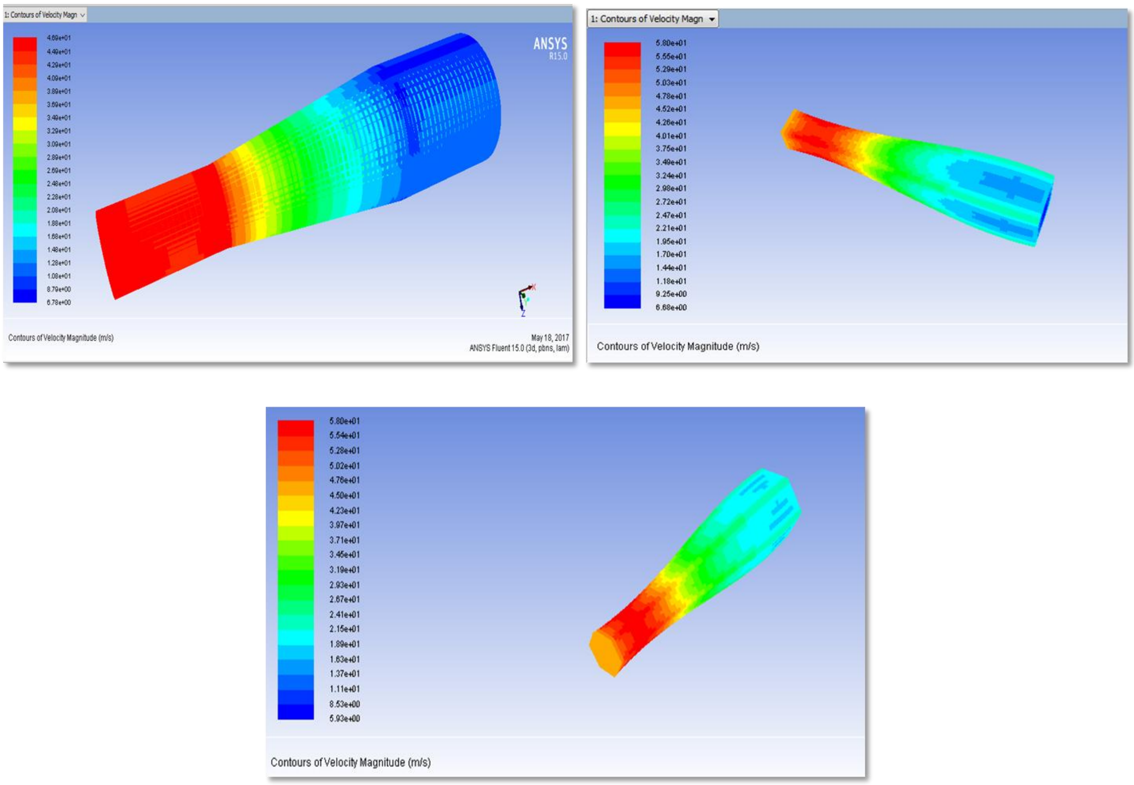
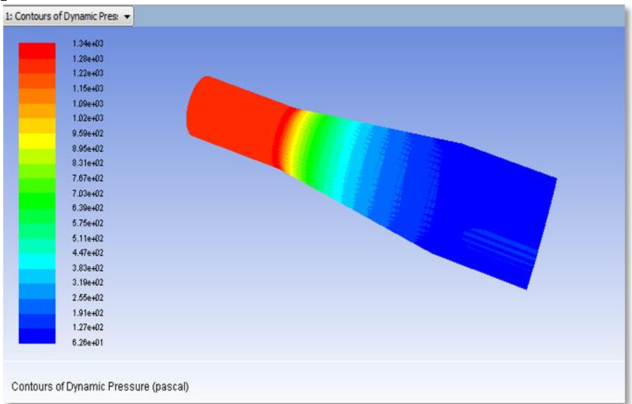


Fig. 8 Velocity Contours for the Exhaust Diffusers

The Temperature Contours for the Exhaust Diffusers is as shown in the Fig. 9



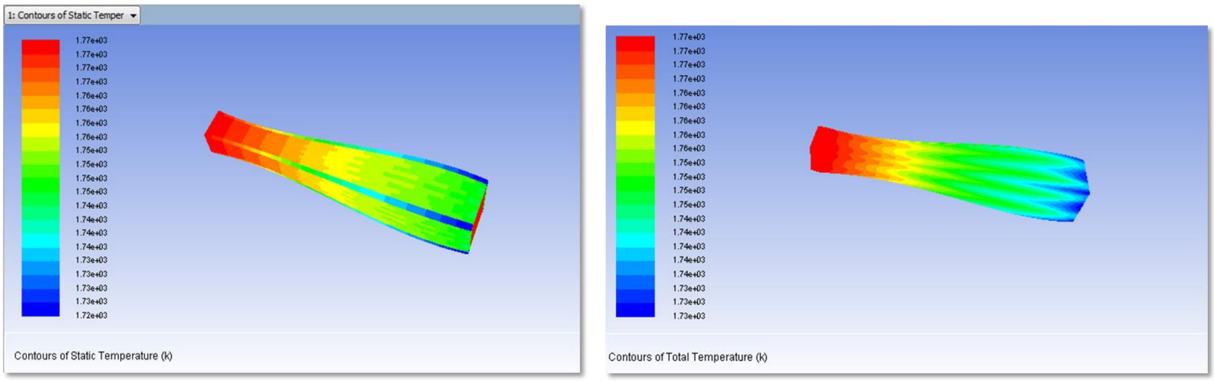


Fig. 9 Temperature Contours for the Exhaust Diffusers

V. THERMAL ANALYSIS IN FEA

The finite element method has become a powerful tool for the numerical solutions of a wide range of engineering problems. It has developed simultaneously with the increasing use of the high-speed electronic digital computers and with the growing emphasis on numerical methods for engineering analysis. This method started as a generalization of the structural idea to some problems of elastic continuum problem, started in terms of different equations or as an extrinum problem.

F.E.A is a way to deal with structures that are more complex than dealt with analytically using the partial differential equations. F.E.A deals with complex boundaries better than finite difference equations and gives answers to the 'real world' structural problems. It has been substantially extended scope during the roughly forty years of its use.

F.E.A makes it possible it evaluate a detail and complex structure, in a computer during the planning of the structure. The demonstration in the computer about the adequate strength of the structure and possibility of improving design during planning can justify the cost of this analysis work. F.E.A has also been known to increase the rating of the structures that were significantly over design and build many decades ago. The thermal analysis of the three diffusers are as shown in The Fig. 10

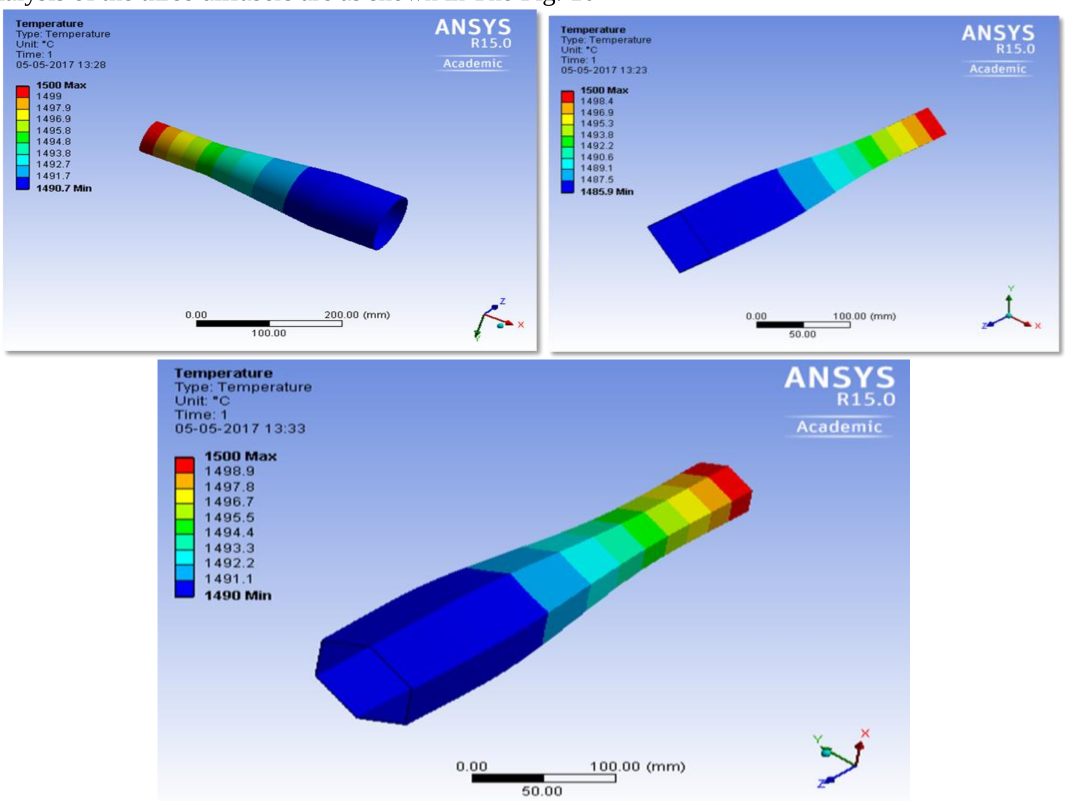


Fig. 10 Temperature affect for the Exhaust Diffusers

Static Pressure:

The results obtained for Static Pressure in CFD are as shown in the below Table. 1

Table 1 Results obtained for Static Pressure in CFD

Shape	Static Pressure (Min), Pa	Static Pressure (Max), Pa
Circular	100082.6	101339.7
Hexagonal	99660.08	101359.0
Square	99700.0	101356.0

Dynamic Pressure:

The results obtained for Dynamic Pressure in CFD are as shown in the below Table. 2

Table 2 Results obtained for Dynamic Pressure in CFD

Shape	Dynamic Pressure (Inlet), Pa
Circular	1343.48
Hexagonal	1760.26
Square	1707.72

Coefficient of Pressure Recovery (CPR):

The Results obtained for Coefficient of Pressure Recovery (CPR) in CFD are as shown in the below Table. 6.3

Table 6.3 Results obtained for Coefficient of Pressure Recovery (CPR) in CFD

Shape	CPR
Circular	0.935
Hexagonal	0.965
Square	0.970

VI. CONCLUSIONS

The following conclusions can be outlined by considering the analysis on different exhaust diffusers. The modeling is done in Creo Parametric 3.0 modeling software. The thermal analysis is performed in Ansys 15.0 workbench. The solutions obtained were then converted to plots and contours using the post processing interface of FLUENT. Computational analysis was performed on various shapes of diffusers and their co-efficient of pressure recovery were calculated using the data obtained. The velocity plots and contours depict an exactly opposite trend, owing to the conversion of kinetic energy into pressure energy. Also, it can be seen that the centerline velocity is higher than the velocity at the boundary due to friction effects at the boundary layer. It was found that the co-efficient of pressure recovery for square diffuser is found to be 0.97 and using this type of diffuser we can improve turbine efficiency and turbine performance.

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.

[12] Ms. B.Krishna Prafulla, Dr. V. Chitti Babu and Sri P. Govinda Rao, "Cfd Analysis of Convergent- Divergent Supersonic Nozzle", International Journal of Computational Engineering Research||Vol, 03||Issue, 5.

[13] Bogdan-Alexandru Belega, Trung Duc Nguyen, "Analysis of Flow in Convergent-Divergent Rocket Engine Nozzle Using Computational Fluid Dynamics", International Conference of Scientific Paper a fases 2015 Brasov, 28-30 May 2015.

