Scientific Journal of Impact Factor (SJIF): 5.71
e-ISSN (O): 2348-4470
p-ISSN (P): 2348-6406

International Journal of Advance Engineering and Research Development

Volume 5, Issue 02, February -2018

THERMAL ANALYSIS OF DIFFERENT CONICAL EXHAUST DIFFUSER USING FEA & CFD

M. Pavan Kumar[1], R. Suresh [2]

[1] Dept. Of Mechanical Engineering, Sri Venkateswara College Of Engineering And Technology, Srikakulam, AP, India
[2] Dept. Of Mechanical Engineering, Sri Venkateswara College Of Engineering And Technology, Srikakulam, AP, India

Abstract — *CFD is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyse that involve fluid flows. In this thesis CFD analysis of flow within diffuser of different cross sections square, circular and hexagonal has been performed. The analysis has been performed according to the shape of diffuser and keeping the same input conditions. Our objective is to investigate the best suited diffuser which gives high co-efficient of pressure recovery.*

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 (Workbench 15.0). The thermal analysis is performed to check for the better material among Stainless steel, Al-alloy and Copper alloy basing on results that include temperature distribution and heat flux distribution. 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- *Pressure recovery, Diffuser, Computational Fluid Dynamics.*

I. INTRODUCTION

A diffuser, in an automotive context, is a shaped section of the car under body which improves the car's aerodynamic properties by enhancing the transition between the high-velocity airflow underneath the car and the much slower free stream airflow of the ambient atmosphere. It works by providing a space for the under body airflow to decelerate and expand (in area, as density is assumed to be constant at the speeds that cars travel) so that it does not cause excessive flow separation and drag, by providing a degree of "wake infill" or more accurately, pressure recovery. The diffuser itself accelerates the flow in front of it, which helps generate down force.

When a diffuser is used, the air flows into the under body from the front of the car, accelerates and reduces pressure. There is a suction peak at the transition of the flat bottom and diffuser. The diffuser then eases this high velocity air back to normal velocity and also helps fill in the area behind the car making the whole under body a more efficient down force producing device by reducing drag on the car and increasing down force. The diffuser also imparts upward momentum to the air which further increases down force. 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.

The concept used in designing a diffuser is the ground effect, that is, to cause a venturi-like effect under the vehicle. Under such a vehicle, there is a nozzle that proliferates the velocity of the air below the vehicle and a throat is formed where the maximum velocity exists and then a component called undertray slows this air back down to free stream velocity. As per Bernoulli's Equation, we know that when the local velocity increases, the local pressure is decreased. Because of this lower pressure under the vehicle and the higher pressure on top, a force called downforce is applied on the vehicle.

The efficiency of the whole component purely depends on the efficiency of the diffuser section. The main role of the diffuser is to slow the air flowing under the vehicle and thus reduce it to the free stream which subsides the drag and increase the overall undertray efficiency. The main moto while designing is to get the highest possible angle without flow separation as in case there is a separation, it may lead to more drag and thus lesser downforce. There are a few more factors that will make a difference in downforce and/or drag.

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.

Manoj Kumar Gopaliya et. al[6] in his paper described that the improvement in performance of S-shaped diffuser with rectangular inlet having aspect ratio 2 & rectangular outlet having area ratio 2 having downstream settling length of 0.5B at Reynolds number 1.37 × 105 with a uniform velocity of 30 m/s due to momentum imparting technique.

In this technique a cylinder of 0.3B diameter is placed at the inflexion plane across the width of the diffuser. Cylinder is rotated at different speed varying from 1500 rad/s to 4000 rad/s to impart momentum to the retarded fluid.

Masafumi Nakagawa et. Al [7] in his paper said that Two-phase flow nozzles are used in the total flow systems of geothermal power plants and in the ejector refrigeration cycle. The purpose of the present study is to theoretically elucidate the characteristics of expansion waves at the outlets of supersonic two-phase-flow nozzles. Two-dimensional basic equations for compressible two-phase flow were derived by incorporating the equation of inter-phase momentum transfer into equations of gas dynamics. In this study, the theoretical analyses were carried out by focusing on momentum-relaxation phenomena in high-speed mist flow.

K.M. Pandey [8] in his paper discussed that Numerical study has been conducted to understand the gas flows in a conical nozzle at different degree of angle using 2 dimensional axi-symmetric models, which solves the governing equations by a control volume method. The nozzle geometry co-ordinates are taken by using of method of characteristics which usually designed for De-Laval nozzle. The present study is aimed at investigating the supersonic flow in conical nozzle for Mach 3 at various degree of angle. The throat diameter and exit diameter is same for all nozzles. The flow is simulated using fluent software.

Vinodkumar S Hiremath [9] in the paper described that the exhaust diffuser is a part of an afterburner of the gas turbine engine which decreases the velocity of gases coming from the low pressure turbine. The decrease in velocity is required to reduce afterburner cold and hot total pressure loss and to increase flame stability. It helps in better flow control and diffusion in exhaust diffuser leading to increased thrust and combustion efficiency. The present work deals with design and analysis of an exhaust diffuser of gas turbine afterburner.

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.

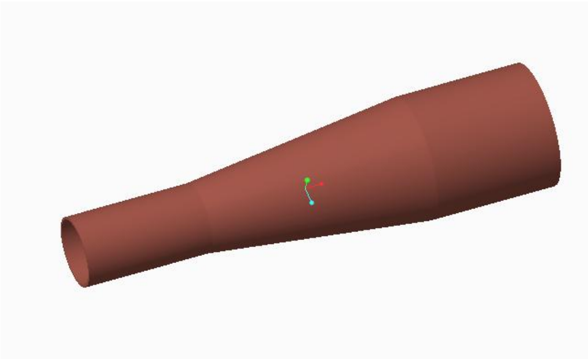


Fig. 1 Model of a circular exhaust diffuser

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

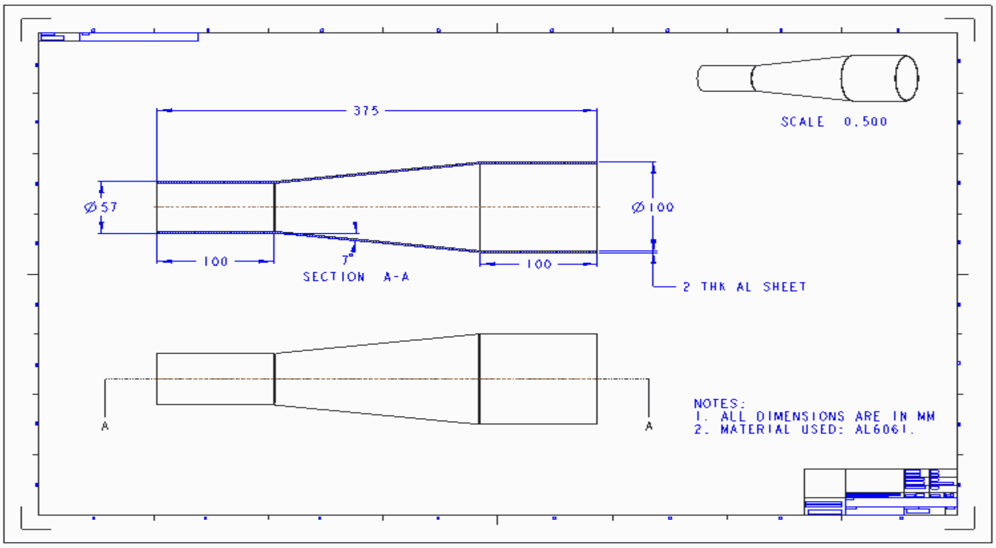


Fig. 1 Drawing Specifications of a circular exhaust diffuser

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.

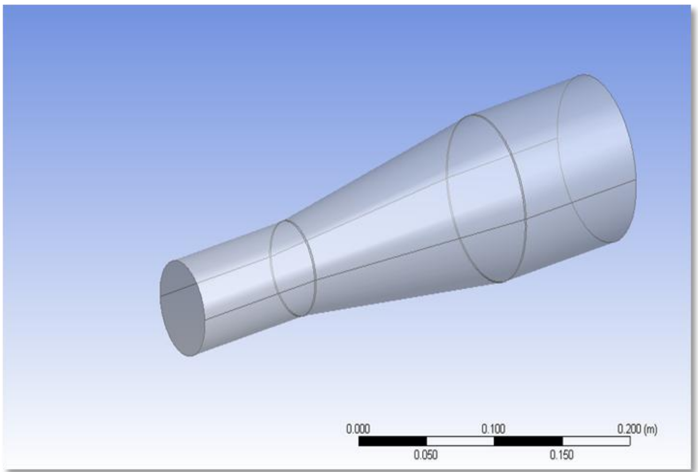


Fig. 3 Geometric model of the circular exhaust diffuser

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

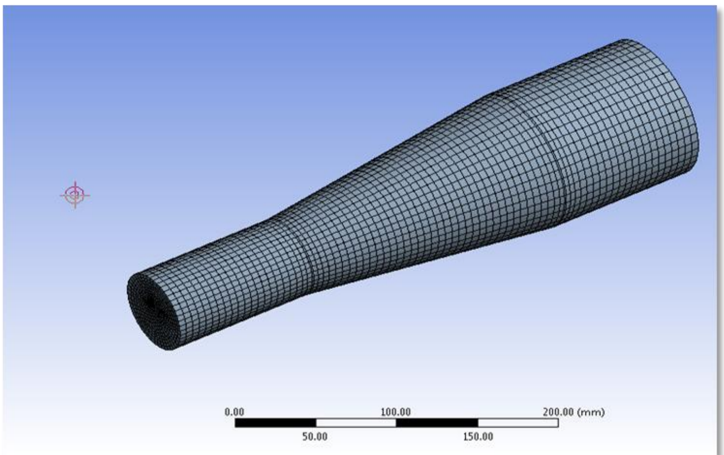


Fig. 4 Meshed model of the circular exhaust diffuser

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

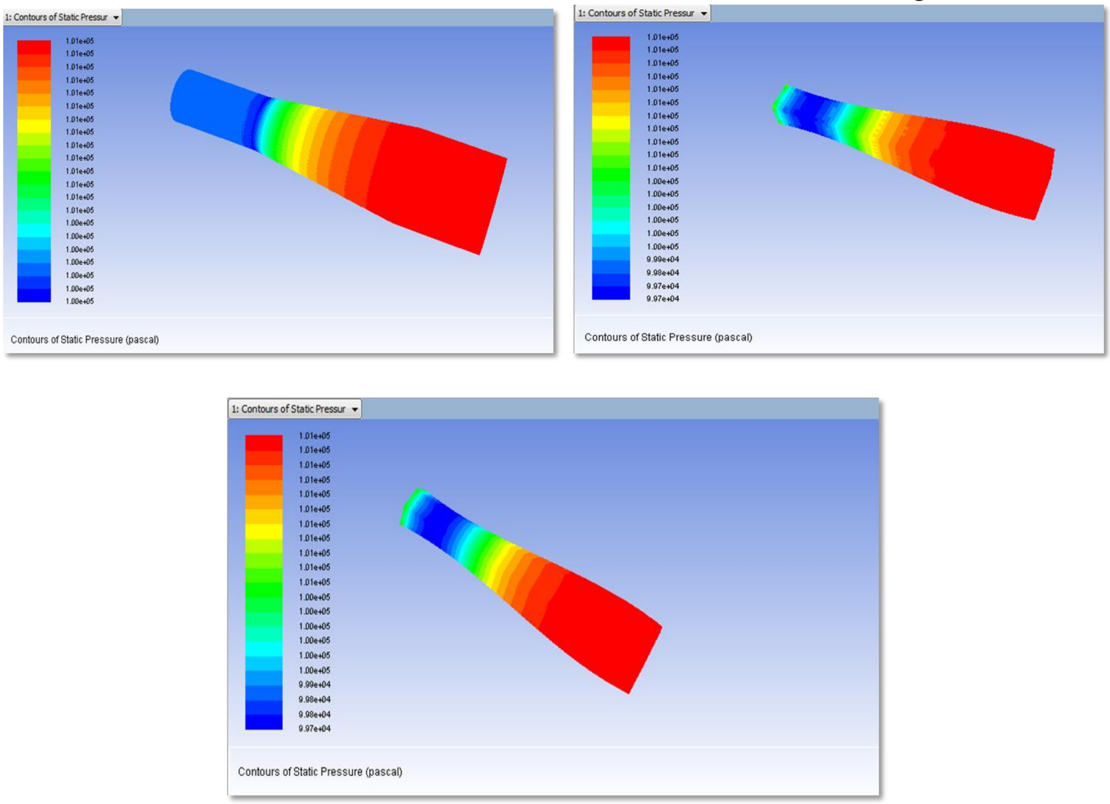
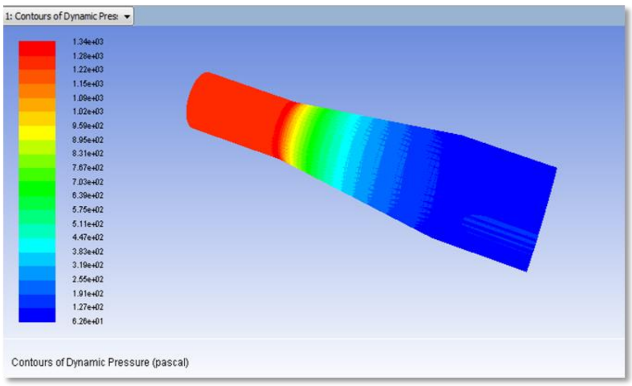


Fig. 6 Static Pressure Contours for the Exhaust Diffusers

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



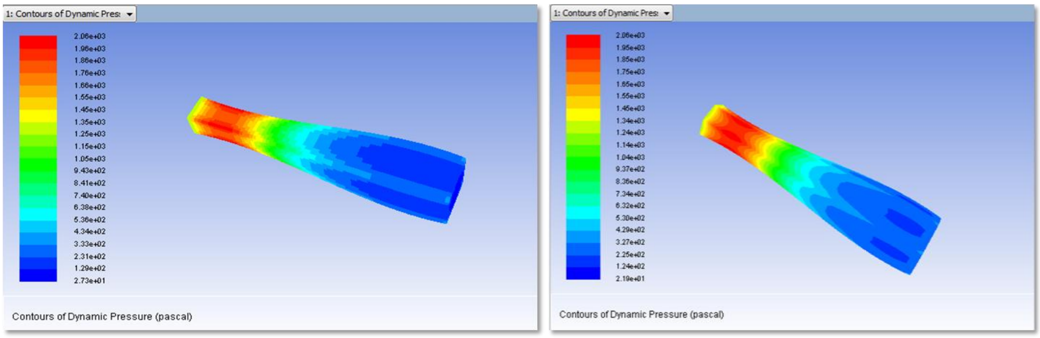


Fig. 7 Dynamic Pressure Contours for the Exhaust Diffusers

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

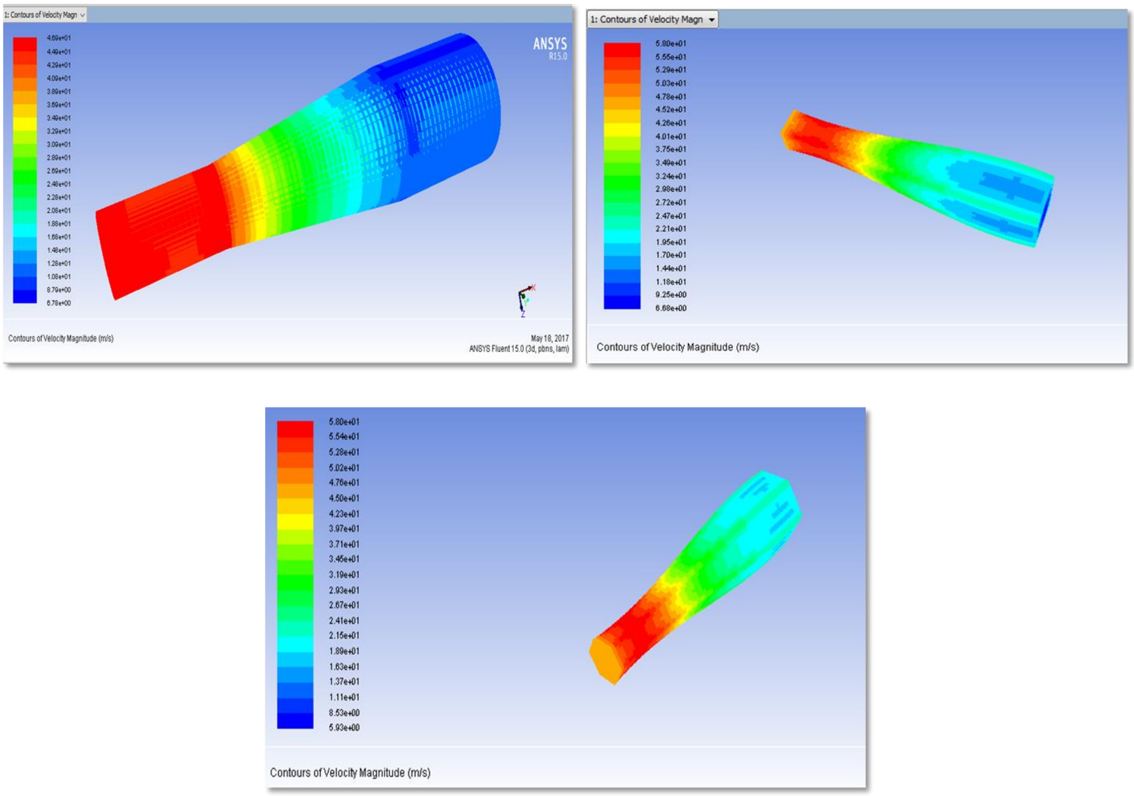
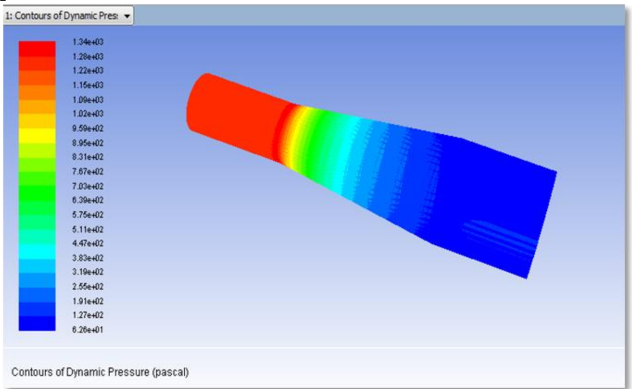


Fig. 8 Velocity Contours for the Exhaust Diffusers

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



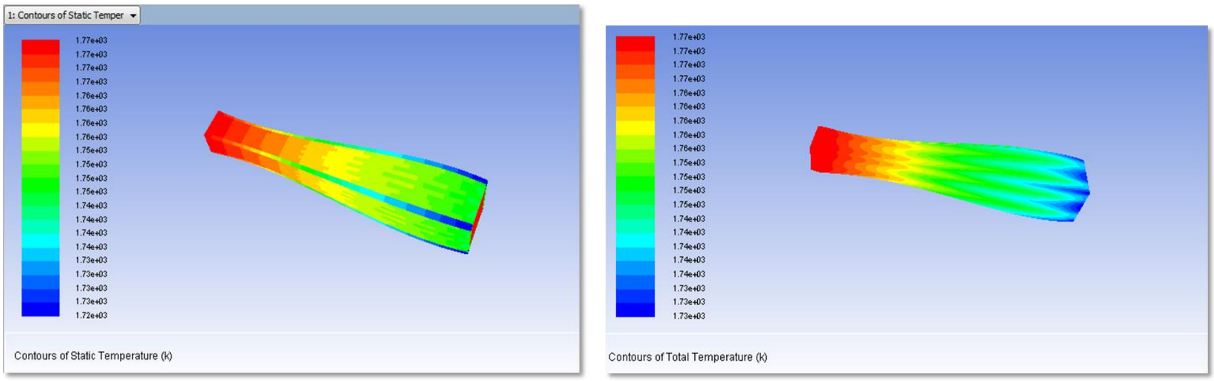


Fig. 9 Temperature Contours for the Exhaust Diffusers

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

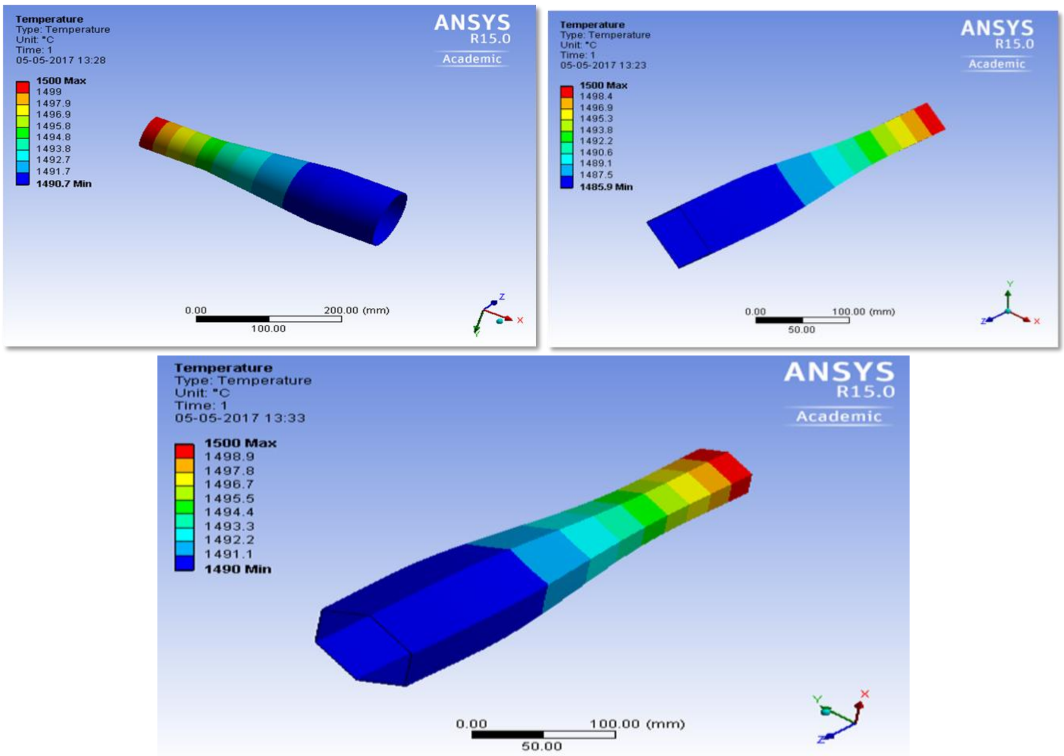


Fig. 10 Temperature affect for the Exhaust Diffusers

Static Pressure:

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

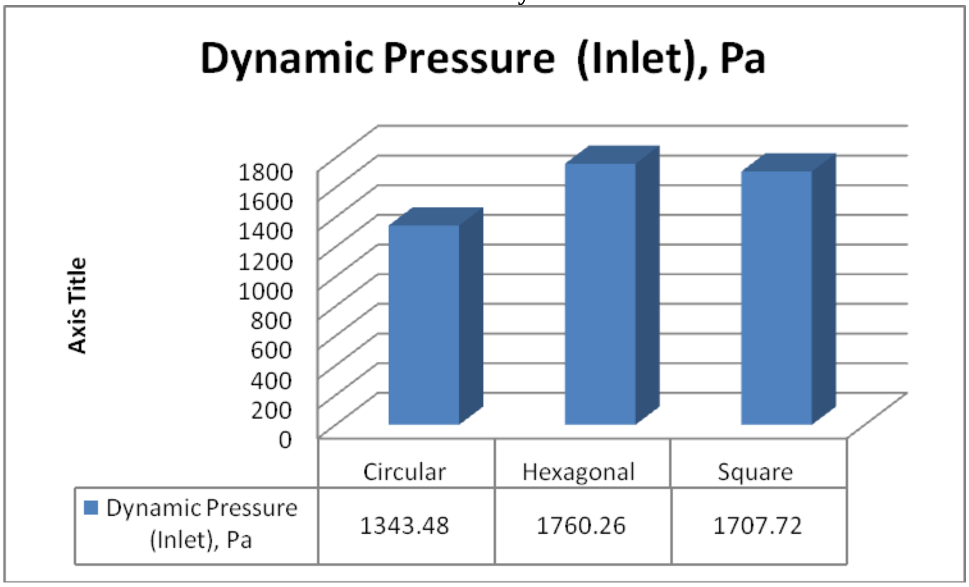
Table 1 Results obtained for Static Pressure in CFD

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

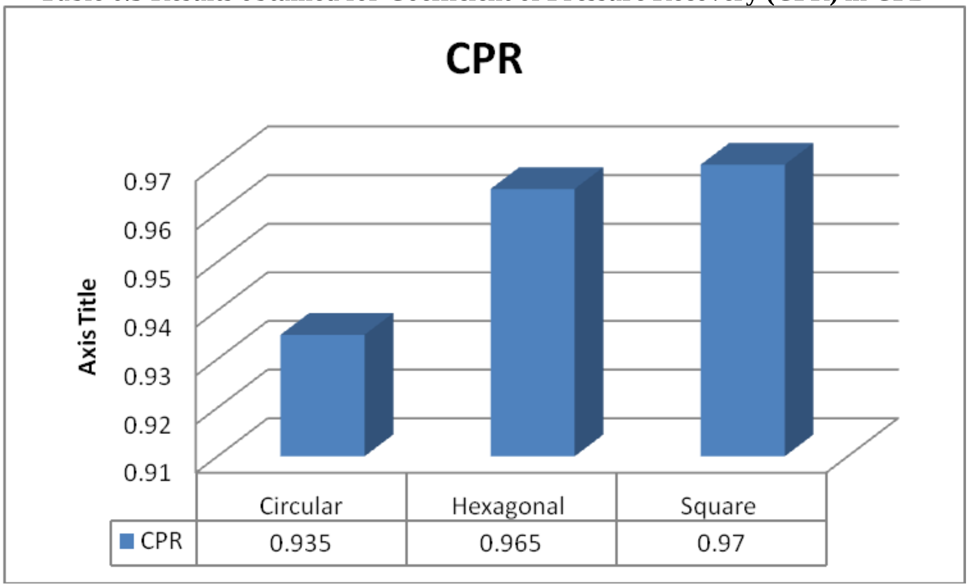
Table 2 Results obtained for Dynamic Pressure in CFD



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



VI. CONCLUSIONS

Computational analysis was performed on various shapes of diffusers and their co-efficient pressures of recovery were calculated using the data obtained. Thermal Analysis was performed on Metals of Aluminum, Copper and Stainless steel with varying shapes on Circular, Square and Hexagonal Diffusers. Taking the Heat flux values into Consideration, This was concluded that as we go on increasing the values of heat flux, Copper is the best suitable metal when compared to other Metals. 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.

