Experimental and CFD Simulation for Improvement in Inlet Duct Design for Existing Compressor

Miss. Bhagat . S. V, Dr . D. M Mate, Prof. S. D. Kathwate
G.H. Raisoni College of Engineering and Management,
Ahmednagar

***Abstract*- During past years, economizing in internal combustion engines has moved the operating conditions of the compressor closer the surge limit. Surge onset depends on the geometry of the compressor duct inlet and it is difficult to predict the flow of streamline. So, in our project our main motive is to develop optimum design of inlet geometries from other shapes i.e. tapered, convergent, divergent, convergent divergent, rectangular, cylindrical from CFD simulation which will improve its recirculation mass flow rate as well as efficiency. Hence, experimental validation will be manufactured with the best optimum design through CFD simulation and comparative analysis will be performed between experimental and numerical results.**

Keywords—ANSYS, CATIA, CFD, Optimum design

I. INTRODUCTION

In the ongoing years, internal combustion engines have advanced because of the innovative improvement typically determined by consistent limitation on toxin discharges. A method that presents an improvement in both poison emanation decreases and expanded execution is the turbo charging of the engines. The expansion in stream thickness and explicit force is utilized to decrease the motor size (spread pattern known as scaling back), boosting the solicitation of low-end torque. So as to accomplish stable conditions at the blower when bringing down the mentioned mass stream rate, as far as possible (flood) should be obviously characterized, being as low as could reasonably be expected. At the point when the blower working point crosses this breaking point, regularly during transient advancements, bothersome wonders show up, for example, brutal pivotal motions of the blower. The funnelling structure of the admission framework can be tuned to kill the contortions that show up, somewhat expanding the effectiveness top. A few creators watched an expansion of the flood edge by putting various geometries at the blower channel. The blower map was moved towards lower mass stream rates by putting a Convergent-Divergent (C-D) spout exceptionally near the inducer. The impact of utilizing drain spaces, procedure otherwise called ported cover or packaging treatment. By and by, every design, likewise of presenting an advantage in the flood edge, may have a positive or negative effect on other blower worldwide factors. To additionally improve proficiency and accomplish higher explicit force, there is a developing pattern toward multistage turbochargers. Blower admission geometries are in every case complex in useful applications as a result of constrained bundling space and these geometries greatly affect the blower execution and stream soundness. Under the pattern of motor scaling back, the establishment space is decreasing and littler, bringing about an increasingly smaller pressure framework design, and the impact of admission geometries on the blower execution, clamor and stream solidness is additionally exacerbated. Scientists watched the exhibition of a pressure framework with an unequivocally bended delta pipe by the CFD strategy and found that the pressure framework execution disintegrated altogether at high mass streams contrasted with the admired apparatus condition (the pressure framework with a straight bay funnel). To kill the negative impact of the curve inlet pipe on the blower solidness and execution.

II. LITERATURE REVIEW

ZhenzhongSuna et.al [1], In this examination it presents the tentatively examination of the stream insecurity of radial compressors with vanes diffuser, and the flimsiness instigation and system at full working extent are introduced and broke down in detail. At low turning speeds, the blower experiences stable state, slow down and profound flood progressively with mass stream rate diminishes and the incitement of the stream unsteadiness is the wave like shakiness aggravation at the impeller inlet. At center turning speeds, the blower encounters stable state, gentle flood and profound flood progressively with mass stream rate diminishes, and the prompting of the stream shakiness is the spike-molded insecurity unsettling influence at the diffuser channel. With mass stream rate diminishes, the blower experiences stable states, slow down and profound flood progressively at low turning speeds. Not withstanding, stable states, mellow flood and profound flood progressively show up at center pivoting velocities, and it is a similar story at high turning speeds. At low turning speeds, it is the wave-like unsteadiness unsettling influence at the impeller bay that initiates the stream shakiness of the blower, while the rule actuation is the spike-molded insecurity aggravation at the diffuser channel at center pivoting speeds. Therefore, examples, actuations and systems of the stream insecurity at full-working condition are introduced in this paper, advancing understandings of the stream flimsiness and enhancements of the dependability of outward compressors.

Sangamesh Bhureet.al [2], In this paper it endeavors to contemplate the impact of back pressure on execution and emanation of diesel engines furnished with EGR and DOC. BPCV is worked physically at three positions, they are 100%, 87.5% and 75% BPCV lifts. The readings are taken in various blends of BPCV lifts and brake torque at 20, 40, 60, and 80 N-
m. The outcomes got shows variety of BPCV lift and brake torque affected on execution of motor, DOC and EGR tasks just as fuel utilization. The NO_x is diminished by 15%; HC and CO are decreased essentially. Be that as it may, there is an expansion in brake explicit fuel utilization (BSFC) and fumes smoke. The expansion in back pressure with brake torque prompts decline in warm proficiency and BMEP, this on account of siphoning misfortune and decrease in explicit warmth valve of fuel that prompts decline in brake influence since motor require extra fuel to defeat this siphoning misfortunes. Expanding back pressure in the fumes framework has expanded the proficiency of DOC gadget by diminishing HC and CO emanation because of expanded time of oxidation. Expanding back pressure, there was decline in BMEP and warm proficiency which prompts decline generally temperature of burning and diminished convergence of oxygen prompts more smoke development. It very well may be reasoned that expanding back pressure by differing BPCV lift under expanding load, assisted with diminishing in HC, CO and NOx emanations.

J. Galindo et.al [3]In this paper it presents the scaling back in internal ignition engines has moved the working states of the blower closer as far as possible. 3D-CFD reproductions are performed utilizing the business code STAR-CCM+ at both close flood conditions and high mass stream rate, concentrating on the stream structures created by every arrangement. Blower simulations with these geometries were made setting two working focuses, one near flood at 30 g/s and one at extremely high mass stream rates at 100 g/s, both at a similar blower speed of 160 krpm. From the merged recreations, worldwide factors were gotten, neighbourhood perceptions in various segment planes were evaluated and the clamour emanation created in both gulf and outlet pipes were estimated, utilizing the Method of Characteristics for the wave deterioration. The outlet pressure waves are additionally affected by the channel geometry, so its examination can't be disposed of in advance for demonstrating the acoustic reaction of the entire gas trade procedure of the motor. A slight commotion decrease is found for the tightened and C-D spout and a huge decrease is created by the merged spout because of the halfway blockage of the delta zone, especially at high massstream rates.

Meijie Zhang et.al [4],in this paper it presents the experimental examinations on the stream flimsiness development of two pressure frameworks with and without an upstream plenum, which speaks to the volume impact of the air channel or the intercooler. The outcomes show that there exists a two-system flood wonder in the pressure framework without the plenum at the medium-and fast lines while this marvel vanishes in the framework with the plenum where the shakiness advancement is clear. Two-system flood is an extraordinary stream wonder in the diffusive blower, in which the pressure framework first experiences gentle flood, at that point gets quiet lastly falls into profound flood with diminishing mass stream rate. This marvel shows up in the pressure framework without the plenum at the 70% and 100% speed lines yet vanishes in that with the plenum. This outcome shows that the instrument of two-system flood depends on the blower itself as well as on the framework setup. At the medium-and fast lines, the profound flood limit of the pressure framework with the plenum movements to the privilege contrasted with that without the plenum, however the mellow flood limit movements to one side. Particularly at the 100% speed line, the steady work go (without gentle flood) stretches out incredibly contrasted with the pressure framework without the plenum. This outcome shows that changing the upstream channel design can possibly broaden the steady work scope of the pressure framework.

III. PROBLEM STATEMENT

In recent years optimization of equipment is trending so, to obtain optimum model from existing component has to be performed very precisely to sustain existing boundary condition. Due to existing geometry of duct of inlet compressor blade design it leads to less recirculation mass flow rate as well as reduction in efficiency. So, to overcome this gap several design including tapered,convergent, divergent, convergent divergent, cylindrical have to be performed in CFD simulation to obtain optimum design for existing geometry.

IV. OBJECTIVES

1. Design of different inlet duct geometries in CATIA software.
2. To perform CFD simulation using ANSYS software of existing centrifugal compressor inlet to improve its recirculation mass flow rate by change in inlet geometries with different inlet duct shape e.g. (tapered, straight, convergent, divergent, convergent divergent, rectangular, cylindrical, square, conical) to improve mass flow rate.
3. To determine the optimum duct shape from CFD simulation for existing compressor blade geometry model to improve its efficiency.
4. Experimental analysis of model for optimum shape and comparing results of CFD simulation.

V. METHODOLOGY

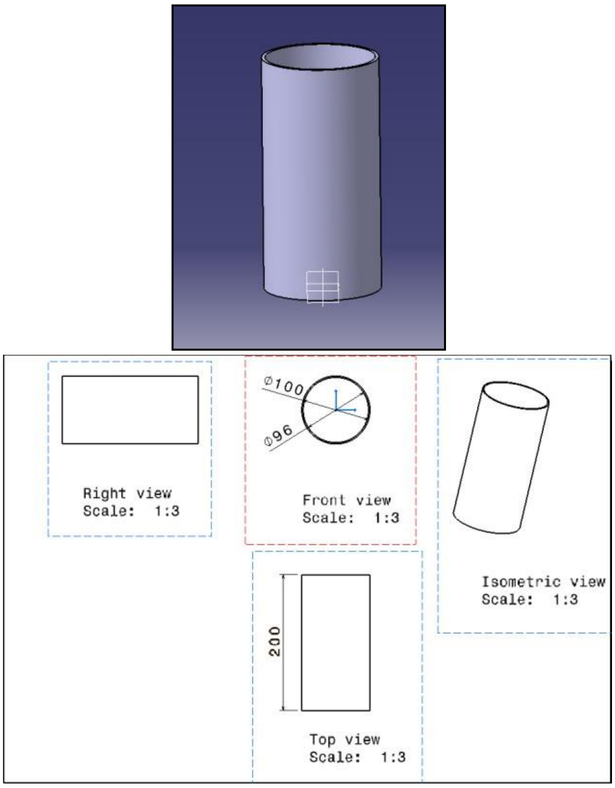
1. Initially research paper is studied to find out research gap for project then necessary parameters are studied

ISSN: 2278-0181

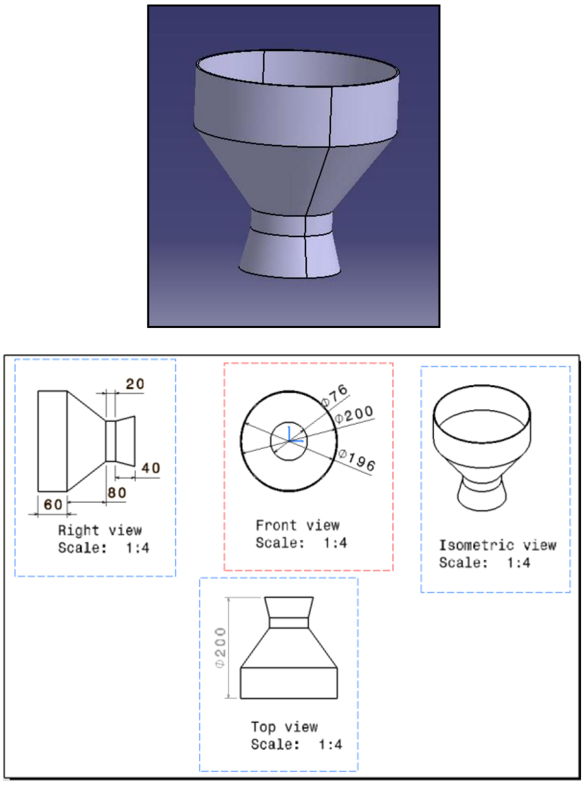
in detail. After going through these papers, we learnt about inlet duct geometries for compressor to enhance efficiency.

2. Research gap is studied to understand new objectives for project.
3. After deciding the components, the 3D Model and drafting will be done with the help of CATIA software.
4. Design of different shape of inlet duct using CATIA software.
5. CFD simulation of different shape inlet duct to calculate mass flow rate, velocity and temperature profile for compressor.
6. In experiment, manufacturing the shape of optimum design obtained from CFD simulation with the use of sheet metal to obtain shape and perform mass flow rate and velocity variation parameters with blower parameters and checking the output results with the help of temperature sensor and anemometer.
7. Validation of results obtained by experiment and CFD simulation.

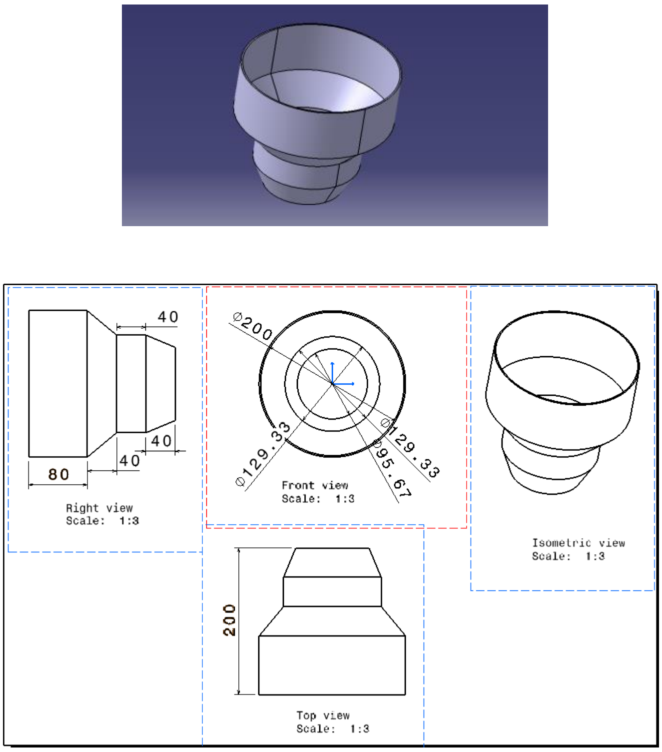
DESIGN OF DIFFERENT SHAPE INLET GEOMETRY



Circular Duct

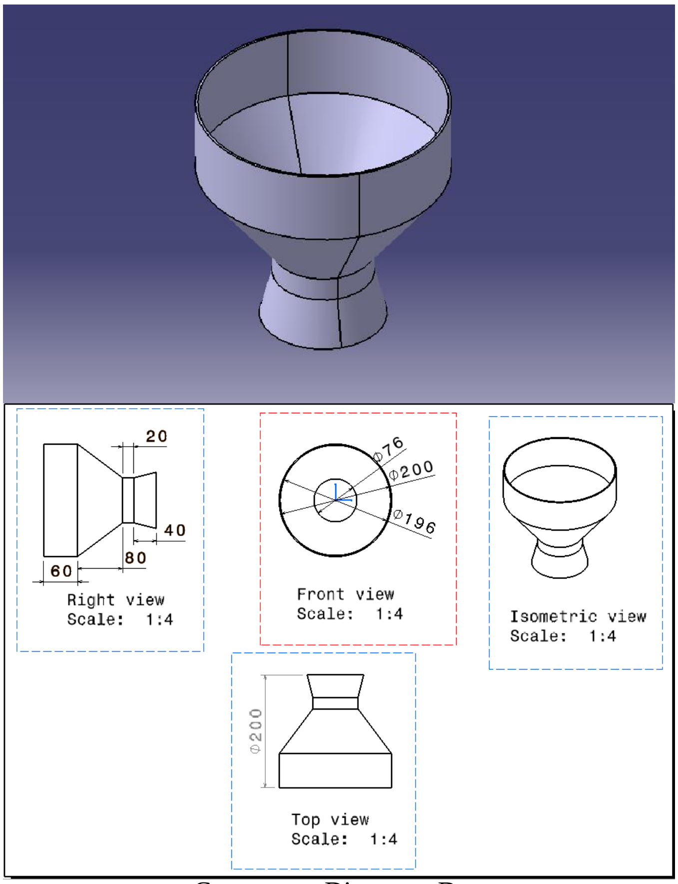


Taper Duct



Convergent Duct

ISSN: 2278-0181
NREST - 2021 Conference Proceedings



Convergent Divergent Duct

Fig.1 CATIA model and drafting of different inlet duct for compressor

For CAD design inlet duct diameter are kept 100 mm as per standard impeller compressor diameter and length of duct is 200 mm only cross section area are varied accordingly standard research papers.

Computational fluid dynamics (CFD) is a branch of fluid mechanics that uses numerical analysis and data structures to analyze and solve problems that involve fluid flows.CFD is now recognized to be a part of the computer-aided engineering (CAE) spectrum of tools used extensively today in all industries, and its approach to modelling fluid flow phenomena allows equipment designers and technical analysts to have the power of a virtual wind tunnel on their desktop computer.

CFD PROCEDURE

- In CFD simulation bounding box is created across blade profile for simulation of velocity and pressure distribution across surface of blade.
- Fine meshing is performed for CFD simulation.
- Named selection is performed in CFD to define air inlet, outlet and blade surface.
- In general box model gravity is defined in perpendicular direction and energy is kept on to perform conservation of mass, momentum and energy equation to solve.
- In viscous model k epsilon, realizable and standard wall function is selected to maintain turbulence flow.
- Inlet velocity is defined as 1.5 m/s.
- Hybrid initialization is performed.
- 100 number of iterations is considered.

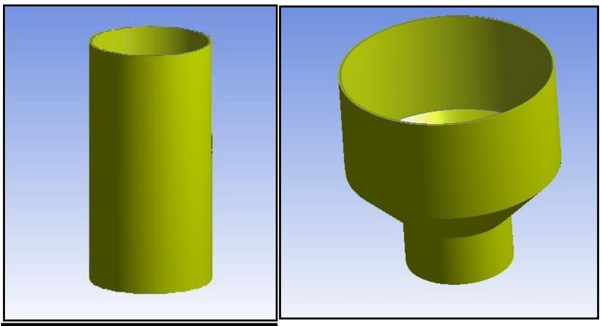


Fig. Geometry imported in ANSYS

Mesh

In ANSYS meshing is performed as similar to discretization process in FEA procedure in which it breaks whole components in small elements and nodes. So, in analysis boundary condition equation are solved at this elements and nodes. ANSYS Meshing may be a all-purpose, intelligent, automated high-performance product. It produces the foremost acceptable mesh for correct, economical metaphysics solutions. A mesh well matched for a selected analysis may be generated with one click for all elements in a very model. Full controls over the options accustomed generate the mesh are accessible for the skilled user who needs to fine-tune it

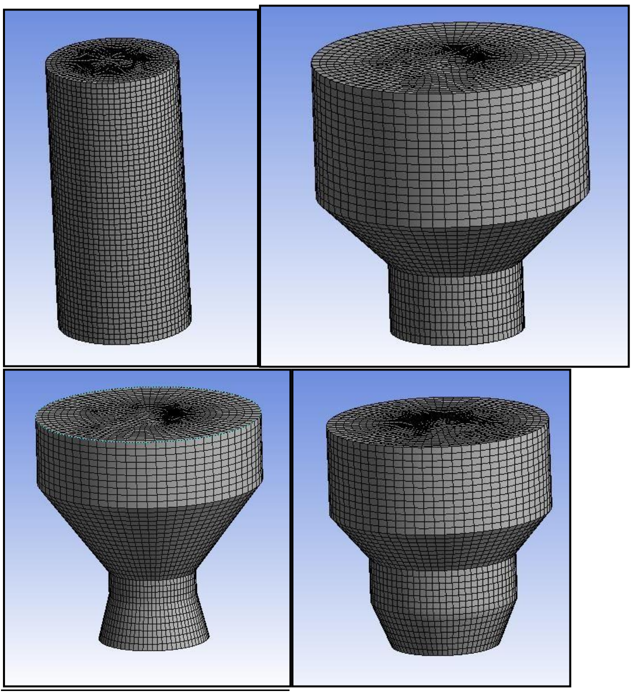


Fig.2 Geometry and meshing of duct geometry

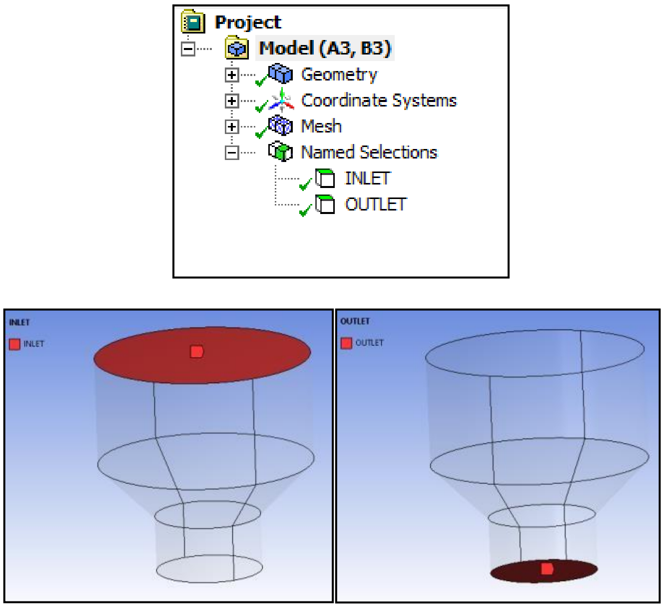


Fig3. Named selection

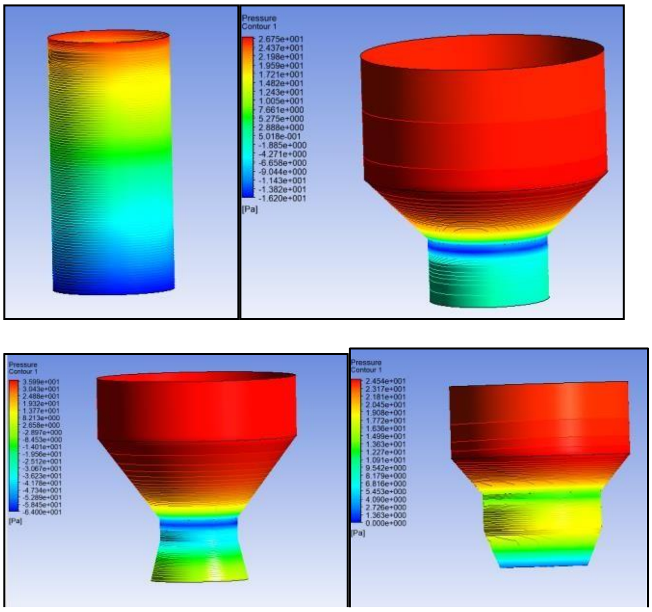
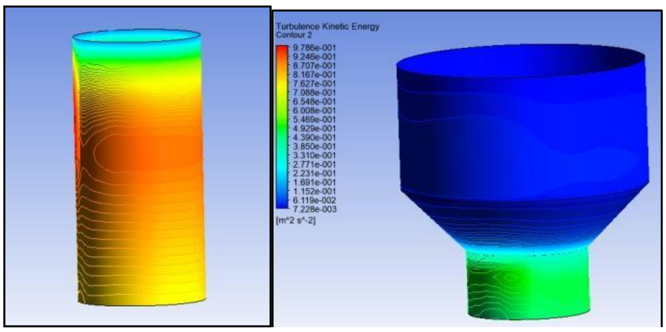


Fig5. Pressure contour of duct geometries



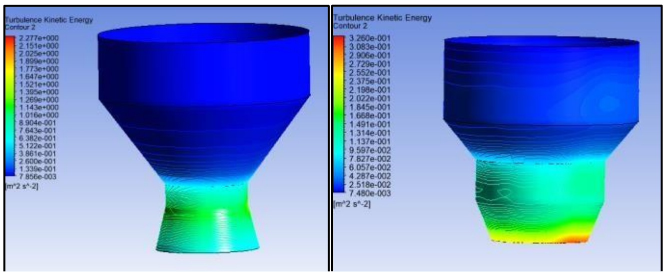


Fig6. Turbulence kinetic energy contour of duct geometries

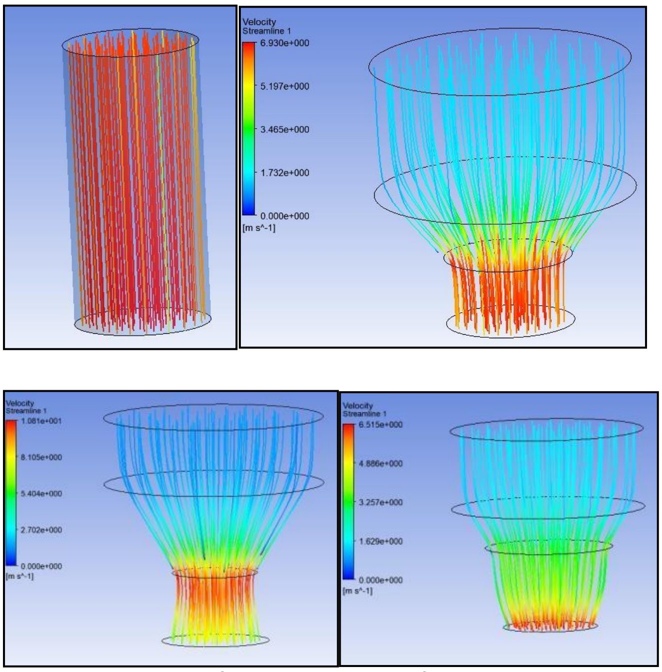


Fig4. Stream flow line contour of duct geometries

EXPERIMENTAL WORK

- Initially design is designed in CATIA software and fixture is assemble to hold the component.
- Pipe of specified dimensions are selected as per standard design along with GI sheet is wrapped to form conical shape with standard shape dimension.
- Rivet joint are preferred over welding to joint components along with M seal to provide leak prof joint.
- For experiment initial velocity as per CFD simulation is considered and outlet velocity is measured by anemometer for validation of CFD results.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
NREST - 2021 Conference Proceedings

Fig.7 Experimental setup

Fig8. Experimental testing result

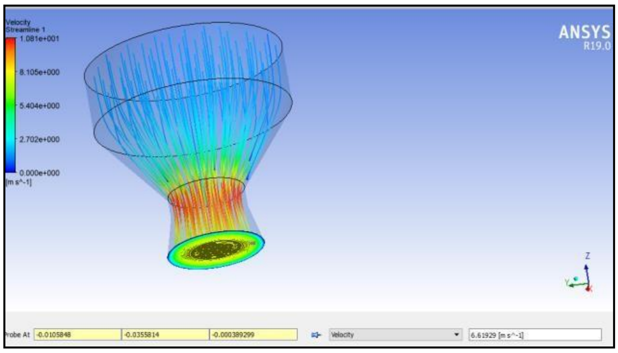
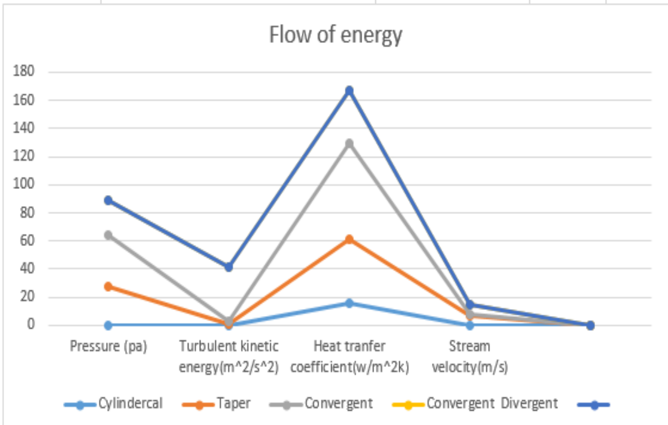


Fig9. CFD simulation result

Result and Discussion



It is observed that for 8 m/s for both experimental and CFD simulation outlet velocity is around 6.7 and 6.61 m/s respectively.

CONCLUSION

1) In present research CFD simulation for different duct namely (cylindrical, convergent, tapered, convergent - divergent) duct are performed to obtain pressure drop, heat transfer coefficient, turbulence kinetic energy to optimize the existing duct design to improve efficiency.
2) It is observed from simulation that duct with shape convergent-divergent have better heat transfer coefficient along with pressure drop compared to cylindrical and other shape.
3) In convergent – divergent turbulence kinetic energy is also improved as 2.2 w/m^{2}K which is useful for recirculation of mass flow rate.
So, in present research convergent divergent duct was optimized design so it was manufactured and experimental testing were performed.

ACKNOWLEDGMENT

I extend my sincere thanks and deep gratitude to my project guide .Dr. D. M .Mate and Co Guide Prof. Kathwate S.D. whose interest and guidance help me to complete the work successfully. I also thanks to Asst .Prof .Kardile S.K.for his valuable advice and guidance. I also like to thank Hon'. Principal Dr. Jayraman I also extended my heartfelt thanks to all the people for their help directly and indirectly to complete our assignment.

REFERENCES

[1] Experimental investigation of instability inducement and mechanism of centrifugal compressors with vaned diffuser, ZhenzhongSuna, XinqianZhenga,, Tomoki Kawakubob, Applied Thermal Engineering 133 (2018) 464–471

[2] Effect of Exhaust Back Pressure on Performance and Emission Characteristics of Diesel Engine Equipped with Diesel Oxidation Catalyst and Exhaust Gas Recirculation, Sangamesh Bhure, Bhure. 2018. Int. J. Vehicle Structures & Systems, 10(3), 199-203

[3] Analysis of the impact of the geometry on the performance of an automotive centrifugal compressor using CFD simulations J. Galindo, A. Gil, R. Navarro, D. Tarí, Applied Thermal Engineering 148 (2019) 1324–1333

[4] Experimental investigation of the flow instability of a compression system with an upstream plenum, Meijie Zhang, Xinqian Zheng, Zhenzhong Sun, Experimental Thermal and Fluid Science 102 (2019) 406–420

[5] K. Ekberg, L. Eriksson, Improving fuel economy and acceleration by electric turbochargercontrol for heavy duty long haulage, IFAC-PapersOnLine 50 (1) (2017) 11052– 11057, 20th IFAC World Congres.

[6] A. Romagnoli,W.-I. Wan-Salim, B. Gurunathan, R. Martinez-Botas, J. Turner, N. Luard, R. Jackson, L. Matteucci, C. Copeland, S. Akehurst, A. Lewis, C. Brace, Assessment of supercharging boosting component for heavily downsized gasoline engines, in: I.O.M. Engineers (Ed.), 11th International Conference on Turbochargers and Turbocharging, Woodhead Publishing, Oxford, 2014, pp. 13– 26.

[7] G.M. Jahangir Alama, Md. Mamunb, "Investigation of the aerodynamic characteristics of an aerofoil shaped fuselage UAV model", Procedia Engineering 90 (2014) 225 – 231Dr.Yadavalli Basavaraj, Manjunatha.T.H, *Design Optimization of Automotive Engine Mount System*, International Journal of Engineering Science Invention Issn (Online): 2319 – 6734, Issn (Print): 2319 – 6726, Volume 2 Issue 3, March. 2013