International Journal of Engineering and Advanced Technology (IJEAT)
ISSN: 2249-8958 (Online), Volume-9 Issue-2, December, 2019

Numerical Analysis of Intake Valve of CI Engine

Pankaj R Pardeshi, K K Dhande, Gorakh P Bhagat, Vikram S Suvarnkar, Vijay K Javanjal

Abstract: ***Internal combustion engines are considered as complex system. The important aspect of any engine is power output which, largely depends upon the proper combustion of air fuel mixture. The aim of this research work is to outline the improve performance of Four stroke Compression Ignition engine by modifications in Inlet manifold geometryThe study is performed on Kirlosakr CI engine. At beginning design of existing inlet valve was studied and CAD drawing is prepared in CATIA V5 Software. Based on existing design and work done by various researchers new modified inlet valve with addition of small plate, is designed in CATIA V5 Software. The numerical investigation is performed using Ansys CFX solver in 14.5 In numerical simulations Engine with two ports is considered to be designed to study the movement of air flow***

Keywords : combustion stages , Inlet valve , analysis of internal combustion

I. INTRODUCTION

The Internal combustion engines are generating mechanical energy as an output from the energy obtained by, complete combustion of fuel. The fuels contain chemical energy. Internal combustion engines receives mixture of air and fuel into the cylinder, in which complete combustion of air-fuel mixture takes place and exhaust gases are given out through outlet port. The power generation process occurs in internal combustion engine in relation with air –fuel mixture which also called as charge. The inward ignition motor consists are of Petrol or Gasoline (spark type), which operates on the Otto cycle, Compression ignition (diesel engines) which operates on the diesel cycle. The aforesaid two engines are having advantage of simple design, high power to weight ratio which makes them more reliable in the various applications varying from automobiles, aircrafts and also in ships. In earlier days heat engines are used by human being over a period of more than two decades. More than period of 100 years, the conversion of water is performed into steam, then this steam is circulated within the products of combustion (gas) are obtained by burning of the fuel while the work is obtained with the aid of piston cylinder mechanism. In 1822-1900 the first engine is developed by

Lenoir J. in which mixture of air and fuel is coming into the cylinder, when piston moves from its initial position in the engine.

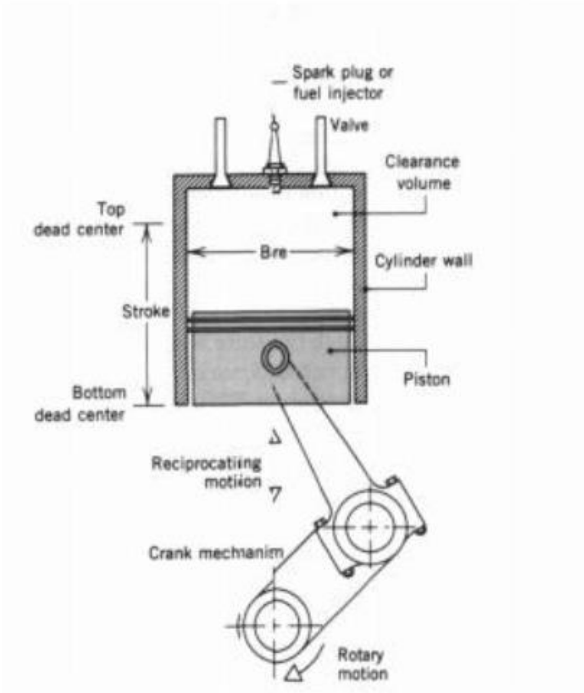

Figure 1.1 Piston and other components of an engine

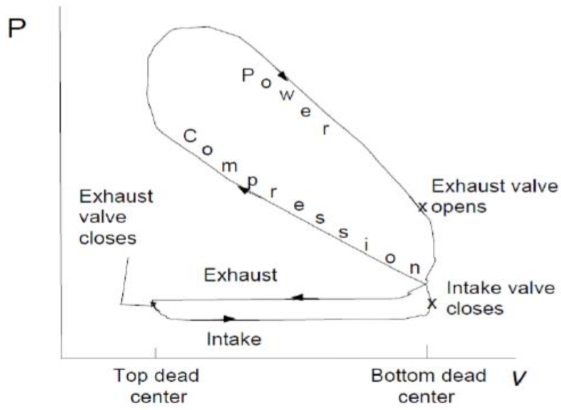

Figure 1.2 P-V Curve

II. REVIEW OF LITERATURE

Andras Horvath et al. (2003) in this work, we contemplate the attributes of the air stream of intake manifold being used for diesel engine through a numerical simulation based on a self-developed code.

Revised Manuscript Received on December 30, 2019.
* Correspondence Author
Pankaj R Pardeshi, Asst. Prof., Mechanical Engineering Department, Dr. D Y Patil institute of Technology Pimpri, Pune
Dr. K K Dhande, HoD, Mechanical Engineering Department, Dr D Y Patil Institute of Technology Pimpri Pune ,
Gorkh P Bhagat, Asst. Prof., Mechanical Engineering Department, Dr. D Y Patil institute of Technology Pimpri, Pune
Vikram S Suvarnkar, Asst. Prof., Mechanical Engineering Department, Dr. D Y Patil institute of Technology Pimpri, Pune
Vijaykumar K. Javanjal, Asst. Prof., Mechanical Engineering Department, Dr. D Y Patil institute of Technology Pimpri, Pune

Several possibilities of the mathematical model of the engineering problem and its numerical solutions are implemented, discussed and partially developed and compared to real physical measurements. As a first order finite volume method, the Vijayasundaram vector division method with local time step control is suitable for accurately calculating the flow properties, namely the flow and rotation coefficients. Precisely, we mean that the calculated and measured quantities differ by 0-0.5% and 0.5-10% respectively, which confirms our numerical model. Then, applying this code and the deformation of the domain, we can increase the flow coefficient by 1% under the condition of a constant number of vortices, which is significant because only small modifications have been allowed.

Steafan Gundmalm et al. (2009) one of the biggest difficulty level in numerical analysis of IC engine is to have defining the correct problem in computational domain. The displacement limits for answer zone formed due to movement of piston and therefore the valves cause a deflection with 37 calculated grid, that ends up in a decrease within the quality of the answer. Based upon this, an older stated mesh methodology has been used in that variety of meshes for circling the whole process of simulation. In every intermesh interval, the interior points of the mesh will be captive to responsible for the limiting movement of the piston and therefore the valves. This method has been imparted to each simple drawing and drawings of actual engines. The aim of this work is to point out its validity for a true and complicated engine design covering an extended simulation gap than before. When a primary detailing of the purpose formation method, investigations were created on the load movement within the cylinder throughout the suction stroke to research the existence of tilting motion. The solution will indicate that no robust tumbling movement is being produced. The result shows that no strong tumbling movement is generated. This happens due to the digital diffusion caused by the lattice structure.

III. COMBUSTION IN ENGINES

A. *Introduction*

Nowadays compression ignition engine are being used as main power source in various engineering applications such as bus, truck, tractors, and locomotives along with number of stationary industrial applications, for electric power generation of small and medium in marine.

The compression ignitions are broadly utilized for all vehicles, mechanical and different territories due to low operating cost. Though with some of the disadvantages such as noise, vibration, smoke, odour and high weight compression ignition engine finds applications in passenger vehicle. Due to aforesaid problems the weight of engine is more. It is observed that the diesel engine has typical smoke and odour which will be due to droplet combustion and incomplete combustion of lean mixture. The typical piston diameter is in the range of 50 to 900 mm, power output are in the range of 2 to 4500 BHP and speed are in the range of 100 to 4000 rpm.

B. Compression Ignition Engine Combustion

Fuel in liquid form is supplied to combustion region which consists of high temperature and high pressure air. Droplet of fuel is surrounded by its own vapour with temperature of 400 to 500 degree and pressure of 25 to 40 bar. The evaporation of liquid is performed with heat absorbed by surrounding air, which lowers the temperature of layer of air surrounding droplet. After sometime gap this temperature is increased by extracting the heat from air. The ignition occurs when air and vapour comes in contact at specific temperature while air fuel ratio is in combustible range. In compression ignition engine fuel is distributed for specific period of time having crank travel from 20 to 45 degree instead of injecting the fuel single time. Turbulence in Spark ignition engine produces unorganized motion of air without specific direction of flow and in which flame is broken up as well as distributed within homogenous mixture of air and fuel. In case of swirl in compression ignition has organized motion of air, well directed which will give fresh air suppler for combustion of fuel and removes the products of combustion which are not required. The delay of ignition period is nothing but starting phase in combustion cycle.

C. *Combustion Stages*

Some measure of fuel is entered amid the start postpone period however not ignited. Now the delay in ignition period is checked from the injection begin to the point from which the p-$^{\theta}$ curve changes to pure compression of air curve. In second stage there will be uncontrolled or vary fast combustion followed by ignition. In second stage the fuel droplet will get sufficient time for spreading over wide area and covered by fresh air while the increase in the value of pressure will be very fast within the delay in ignition. Third stage is also called as controlled combustion stage. As the second stage there will be uncontrolled or vary fast combustion followed by third stage as controlled combustion.

With the maximum temperature of cycle, it is considered that the duration of controlled combustion will finish. The heat energy developed during this stage is around 70 to 80 % of total heat energy of fuel supplied during engine cycle. The aforesaid are the three stages of combustion and the fourth stage is added to this as afterburning or late burning. Fourth Satge is afterburning stage. It is assumed theoretically that after third stage there will be the end of combustion process. But it is observed that the combustion process will remain continue during remaining half of expansion cycle due to improper distribution of fuel particles. Hence this afterburning is known as fourth stage of combustion. The after burn stage period will be considered from 70- 80 degree of crank movement from TOP Dead Center. The heat energy developed during this stage is around 94 to 96 % and 3 to 4 % of l heat energy of fuel is dissipated with exhaust gases.

IV. RESEARCH METHODOLOGY

In present study intake manifold of variable compression ratio (VCR) engine is designed with the help of CATIA V5. CATIA v5 R20 is considered as a one of the good solid modelling software in the family of solid modelling software. The VCR engine cylinder the main dimensions are cylinder diameter or bore and stroke length. The cylinder bore diameter 79.5mm and stroke length is taken as 95.5mm.

Retrieval Number: B3538129219 /2019©BEIESP
DOI: 10.35940/ijeat.B3538.129219
Journal Website: www.ijeat.org

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication

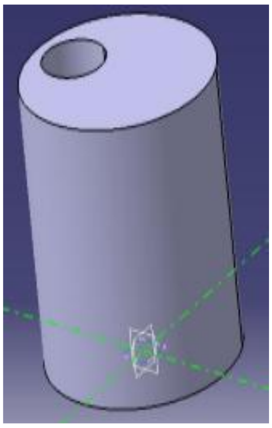

Figure 4.1 Cylinder geometry created in CATIA

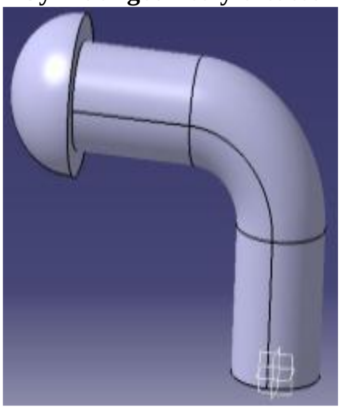

Figure 4.2 Inlet pipe geometry created in CATIA

Figure 4.3 Swell valve geometry created in CATIA

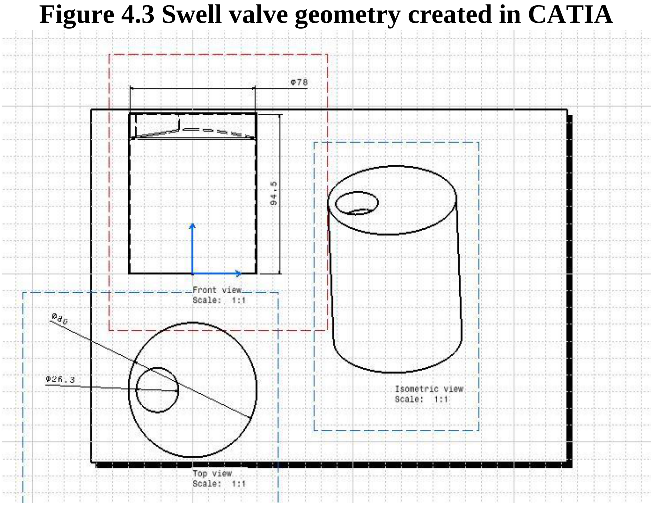

Figure 4.4 Cylinder drawing created in CATIA

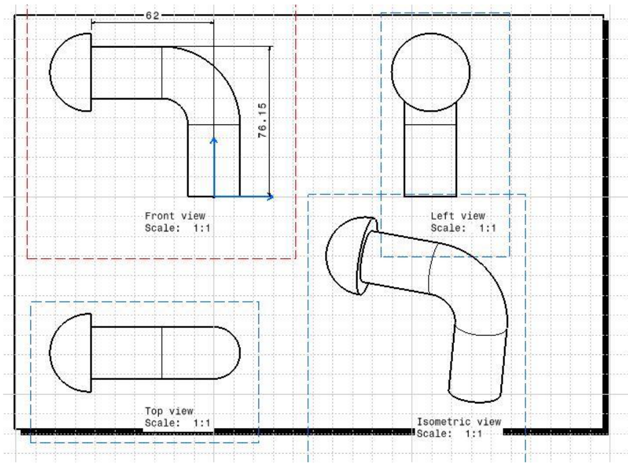

Figure 4.5 Inlet pipe drawing created in CATIA

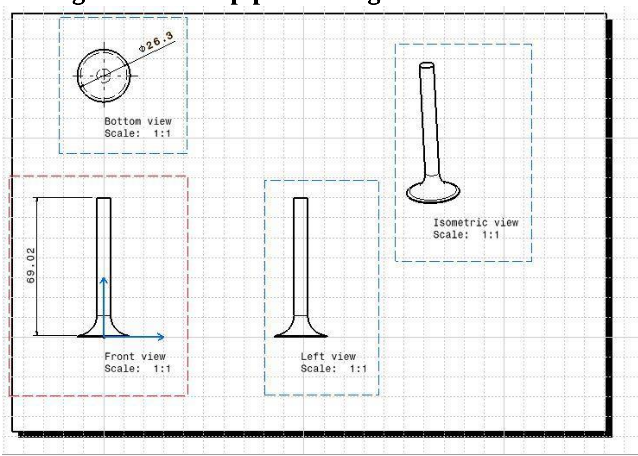

Figure 4.6 Swell valve drawing created in CATIA

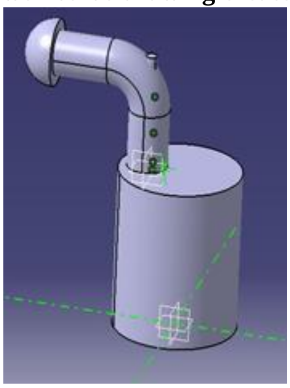

Figure 4.7 Cylinder head part with conventional valve

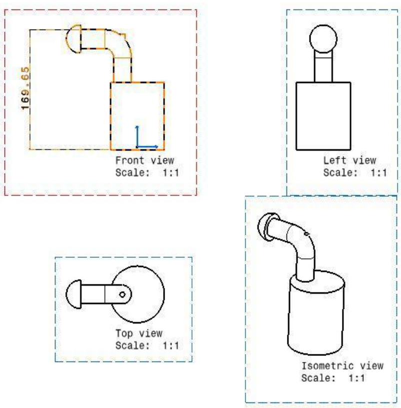

Figure 4.8 Assembly drawing created in CATIA

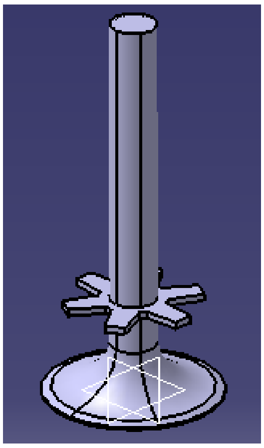

Figure 4.9 Modified Valve in CATIA

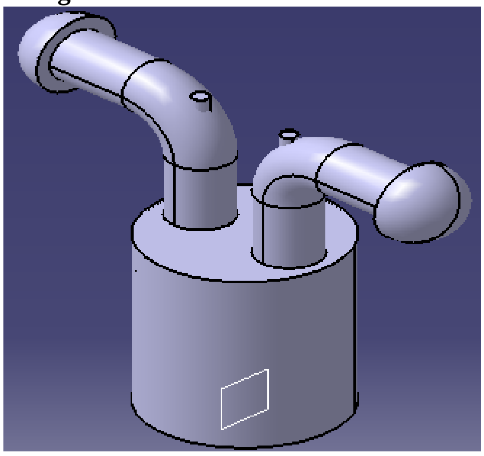

Figure 4.10 Engine geometry after assembly

A. *Importing Mesh*

After meshing, mesh was imported in CFX for further processing. Model selected for analysis is K-epsilon feasible turbulence model show with standard divider capacities. Energy equations are turned on. The calculations are performed by solving compressible Navier-Stokes equation for mass, momentum and energy. Also two equation turbulence model, Realizable $\kappa - \varepsilon$ is used to capture the streams including turn, limit layer under solid antagonistic pressure slopes, partition and distribution. The constant velocity of 10 m/s is given to both inlets. In this study, we have chosen the type of inlet as pressure inlet and applied absolute pressure as 0 pascal to it. And bottom of the wall is chosen as outlet.

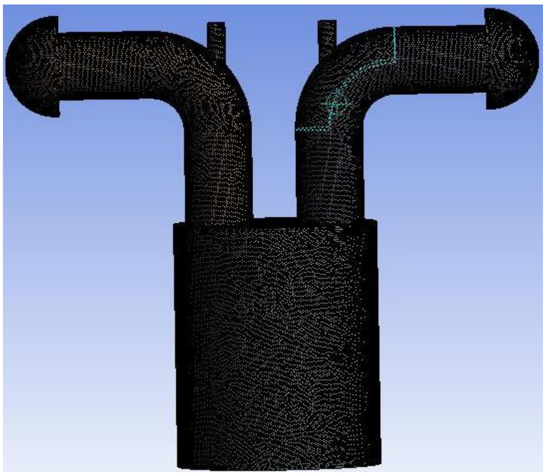

Figure 4.11 Meshing of geometry

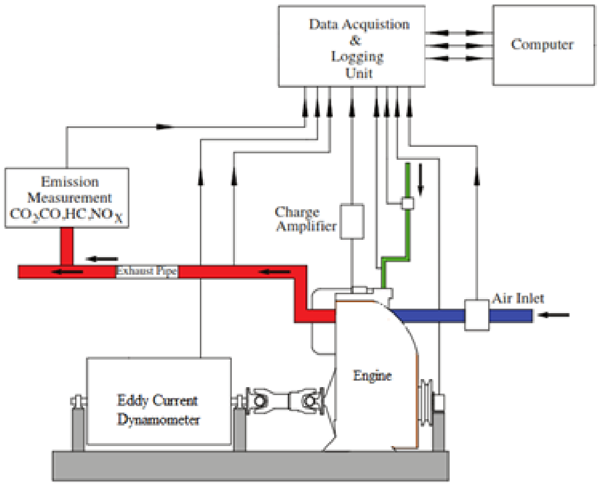

Figure 4.1 Schematic experimental Setup

Figure 4.2 Actual Experimental Setup

V. RESULTS & DISCUSSION

Steady state wind stream computations are performed for six distinctive admission valve lifts viz. 4 mm, 6mm and 8mm for both regular valve and new modified valve configurations

The calculations are performed by solving compressible Naiver-Stokes equation for mass, momentum and energy.

Analysis of existing valve with a valve lifts of 4 mm

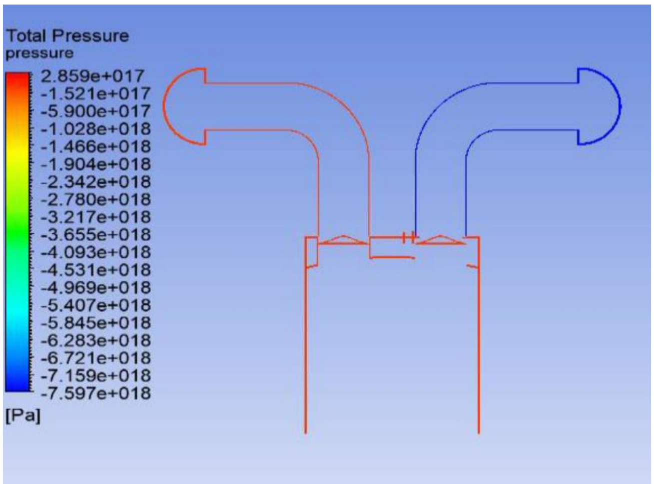

Figure 5.2.1 Pressure in XY Plane

Retrieval Number: B3538129219 /2019©BEIESP
DOI: 10.35940/ijeat.B3538.129219
Journal Website: www.ijeat.org

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication

International Journal of Engineering and Advanced Technology (IJEAT)
ISSN: 2249-8958 (Online), Volume-9 Issue-2, December, 2019

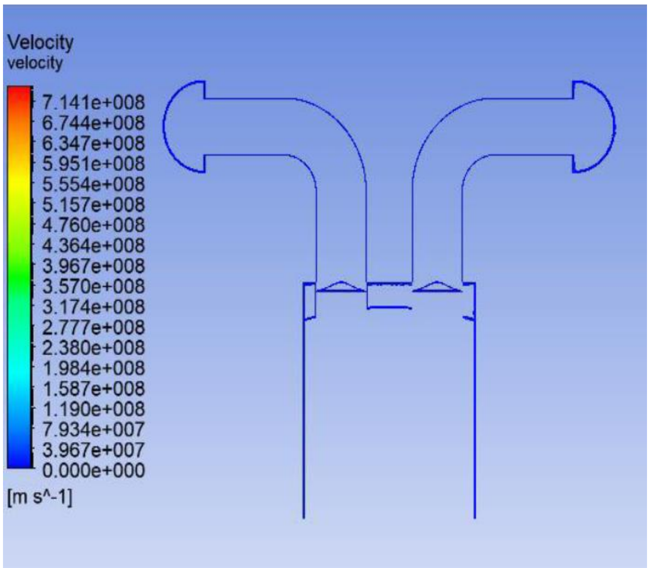

Figure 5.2.2 Total velocity in XY Plane

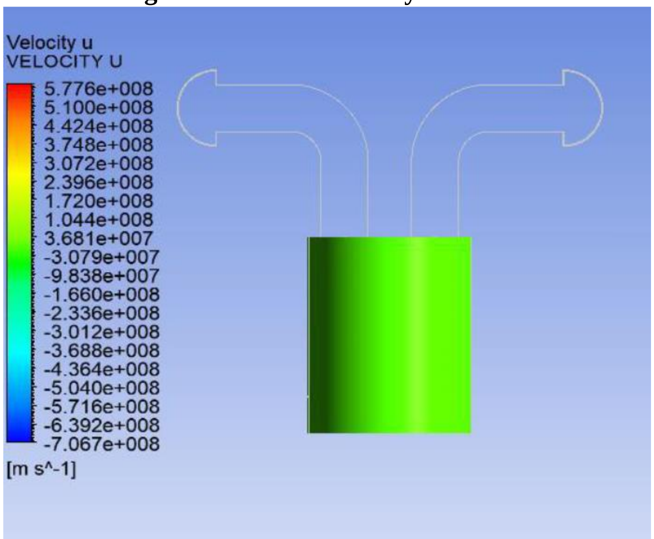

Figure 5.2.3 velocity along u

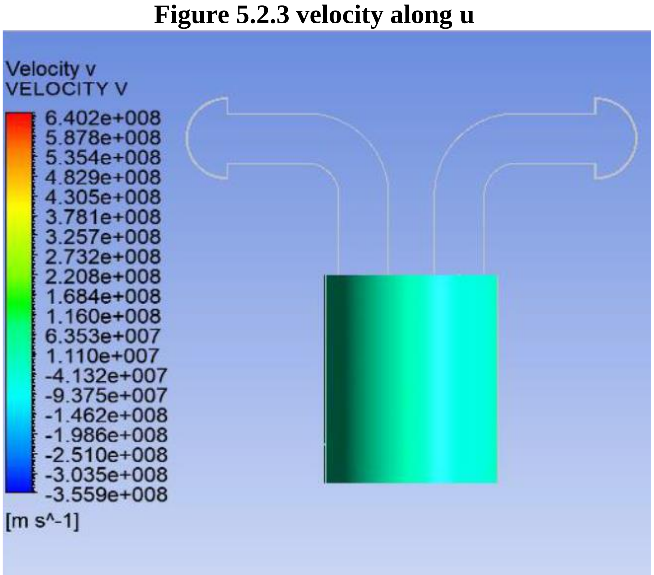

Figure 5.2.4 velocity along v

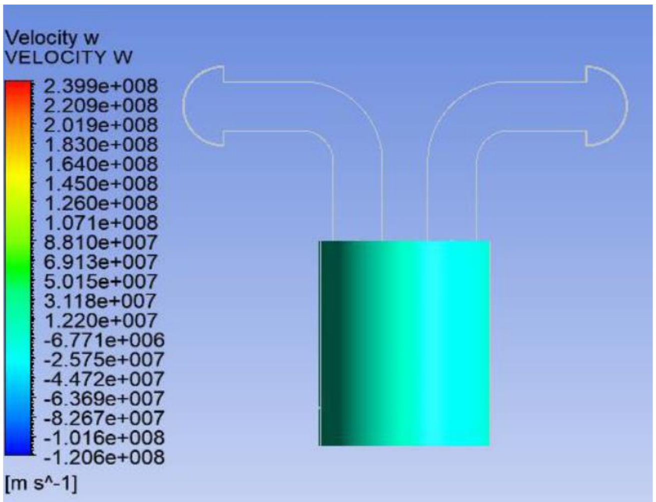

Figure 5.2.5 velocity along w

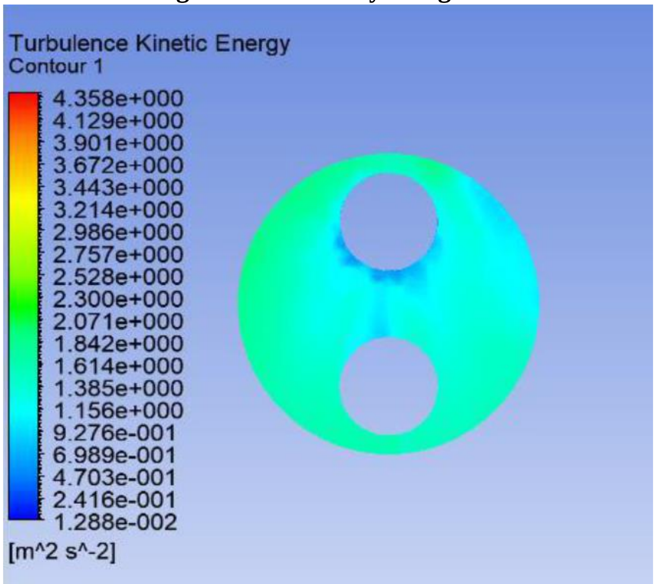

Figure 5.2.6 Turbulent Kinetic energy in XY

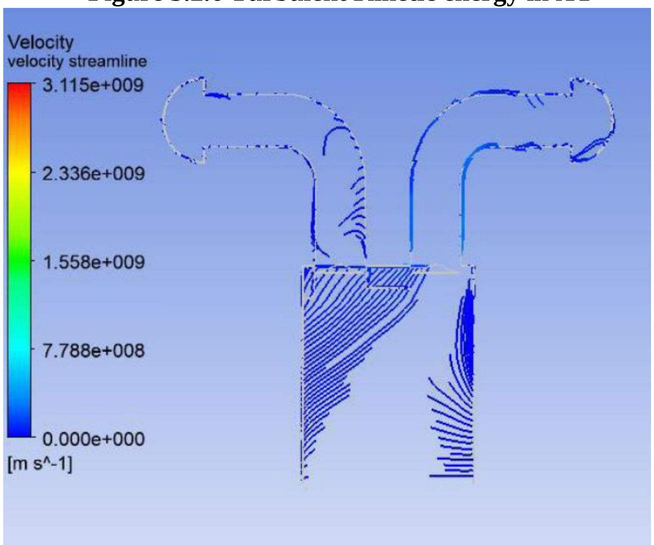

Figure 5.2.7 Velocity streamline in XY

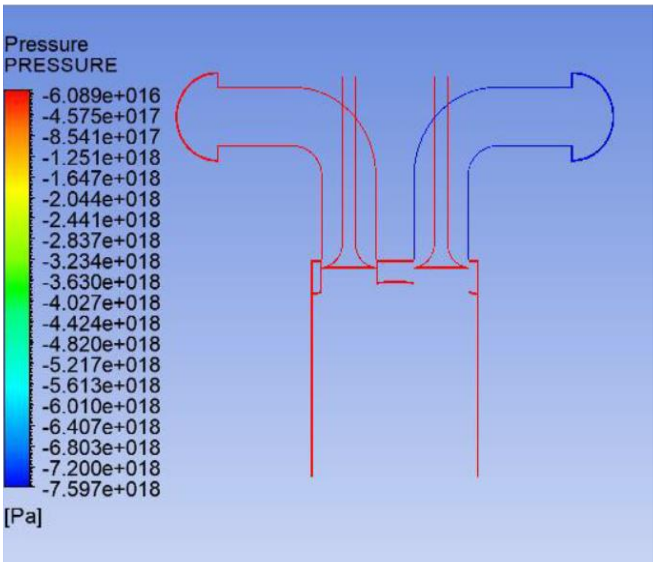

Figure 5.2.8 Pressure in YZ Plane

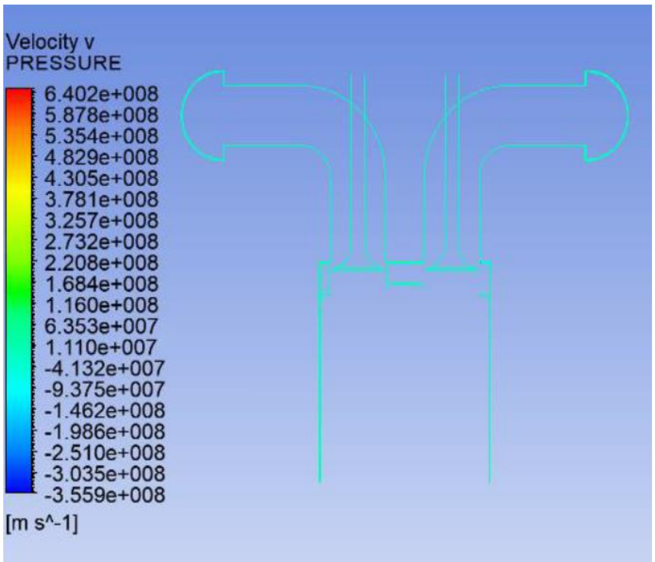

Figure 5.2.11 Velocity in v direction

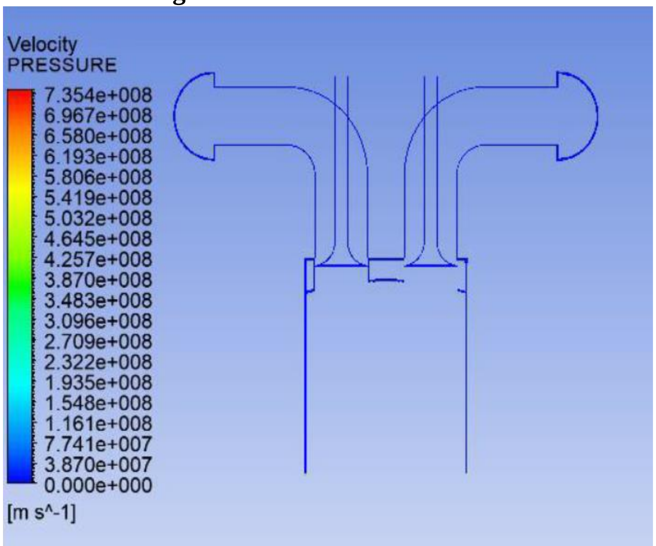

Figure 5.2.9 Total Velocity in YZ Plane

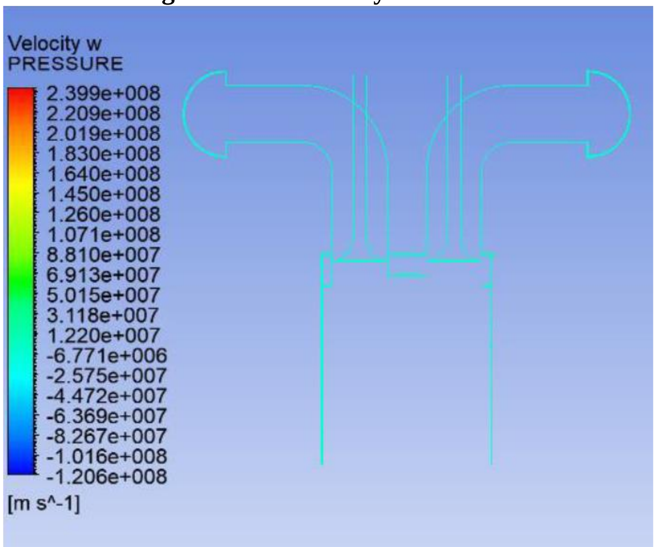

Figure 5.2.12 Velocity in w direction

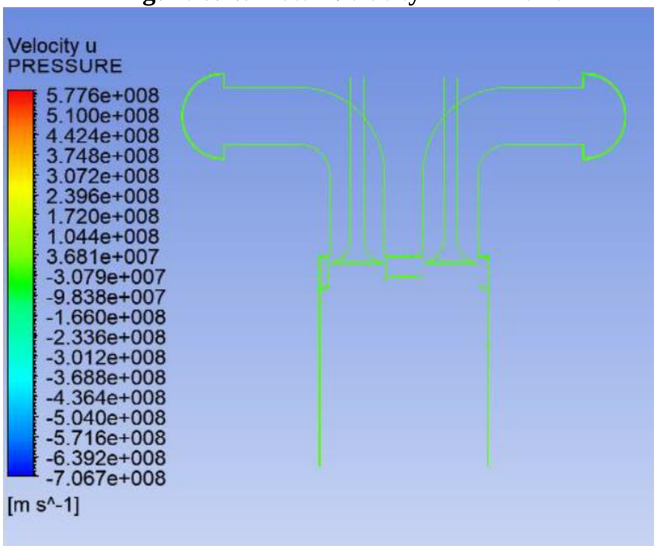

Figure 5.2.10 Velocity in u direction

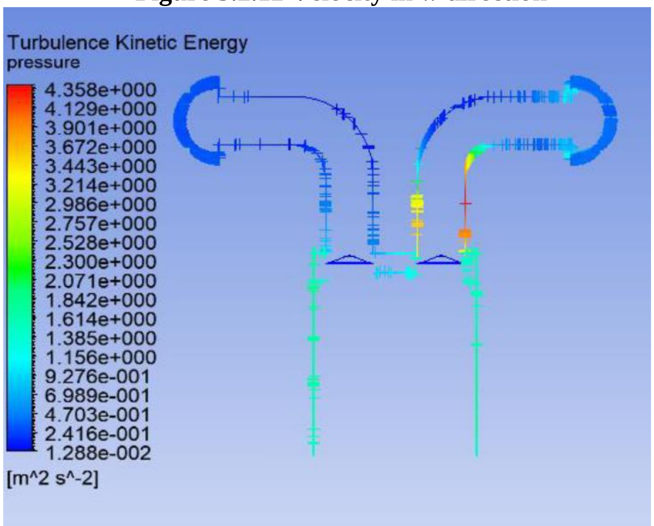

Figure 5.3.6 Turbulent Kinetic Energy XY plane

Retrieval Number: B3538129219 /2019©BEIESP
DOI: 10.35940/ijeat.B3538.129219
Journal Website: www.ijeat.org

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication

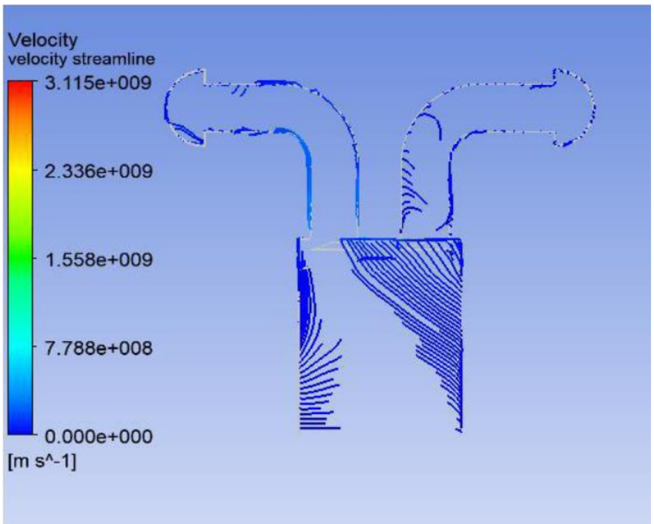

Figure 5.3.7 Velocity streamline XY plane

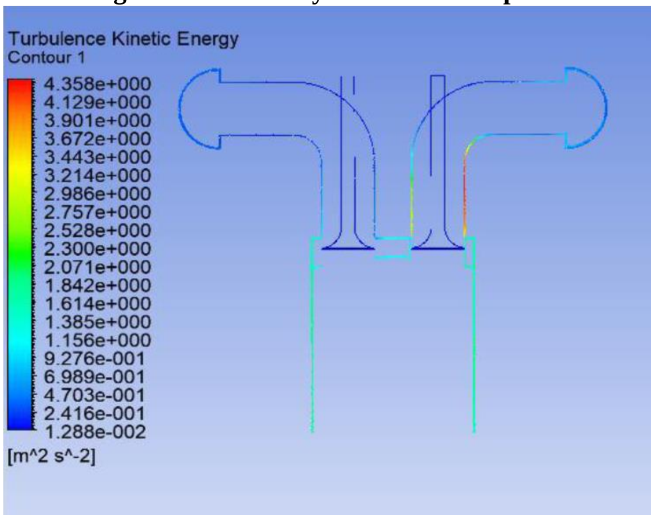

Figure 5.3.13 Turbulent Kinetic Energy YZ plane

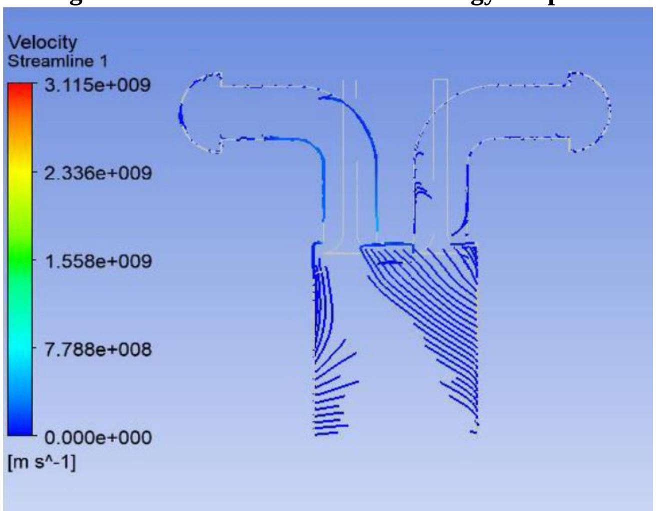

Figure 5.3.14 Velocity streamline YZ plane

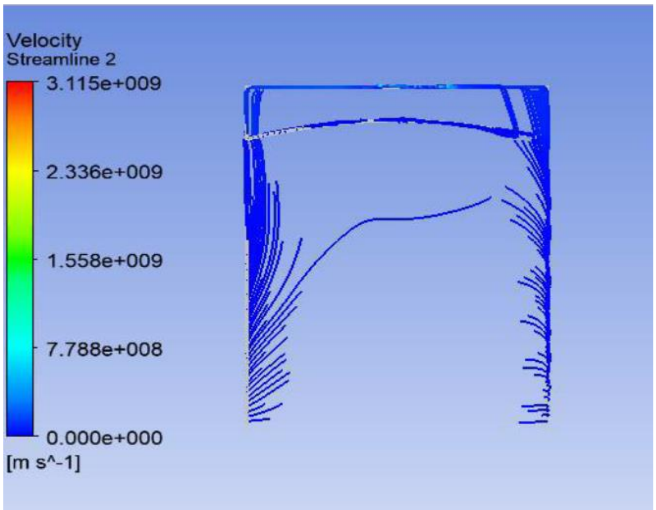

Figure 5.3.21 Velocity streamline ZX Plane

VI. SUMMARY

The research work carried out is about numerical investigation of the intake valve of compression ignition engine. The intake valve of engine is important from the performance point of view because the air coming from intake valve will regulate the combustion process inside the cylinder or an engine. The aim of intake valve is to improve flow of air inside engine within incoming stroke. Conventional engine valves are poppet type of valves which will improve the velocity of air flow up to certain extent but it has limitation due to which the combustion of fuel is not complete. Incomplete ignition of fuel leads to increase in exhaust gas emissions and lower power out of engine. The efforts are made by many researchers, scholars and industrialists for improving the design of conventional intake valve design. As mentioned in literature review the most of the work is targeted towards the improvement of engine performance, reduction of exhaust gas emissions for compression ignition engine, spark ignition engine and duel fuel engines. The work done by researcher is either modification of intake manifold geometry, by varying plenum length, design of adjustable plenum , modification of poppet valve by providing mask and many other types of geometrical modifications are discussed. The aforesaid work is performed wither with the help of computational fluid dynamics approach in which complex partial differential equations are solved, or experimental investigations and both numerical simulation and experimental approach. The experimental or numerical simulation results are giving effect of improved air flow motion on engine performance. Thus in this study the aim is to develop the alternative design to conventional intake valve which will give better air motion to the air while admitted in the engine cylinder during the intake cycle. Thus the primary aim of this research work is to evaluate air flow motion in both the intake valve designs. The present investigation is performed for the study of air jet motion inside incoming manifold of diesel engine. Previous work done by researchers as mentioned in the literature review has been used in the engines with different applications.

The air flow motion investigation will be used by other areas of engine applications such as aircraft, marine and many more. The engine valve modifications will be effective with due considerations to other geometrical aspects. In this study engine used is variable compression ratio engine. Progress of research objectives which has been completed are in start the detailed study of existing intake valve design is completed and its modelling is done in CATIA then the based upon the solid model of existing intake valve a new modified intake valve geometry is designed and having skirt type arrangement. In further progress the air flow motion is studied by means of numerical simulation of the existing intake valve design and modified intake valve with plate. The research approach used in present investigation is of applied research in which immediate solution to the problem being address was found out.

The simulation based approach consists of use software tools such as CATIA for solid modelling of the intake valve configurations and use of computational fluid dynamics approach for virtual evaluation of real time system with actual boundary conditions in order to obtain air flow velocity an pressure values.

The Computational Fluid Dynamics with ANSYS CFX has three fundamental steps

1. Pre-processor

2. Solver and

3. Post-processor

A. Pre-processor

Pre-handling comprises of the contribution of a stream issue to a CFD issue by methods for an administrator benevolent interface and the resulting change of this contribution to a shape reasonable for use by the solver. The client exercises at the pre-handling stage include:

- Definition of the geometry of the district of intrigue: the computational space.
- Grid age the sub-division of the space into various littler subdomains: a network (or work) of cells (or control volumes or components).
- Selection of the physical and concoction wonders that should be demonstrated.
- Definition of liquid properties
- Specification of fitting limit conditions at cells which concurs with or touch the area limit.
- Post– processor: Post-processor is adaptable information representation devices. These include: Domain geometry and lattice show, Vector plots and Contour plots.
- The points of interest of test approach are as take after, the setup comprises of single barrel, four stroke, VCR (Variable Compression Ratio) Diesel motor associated with whirlpool current write dynamometer for stacking.

Impact: The air flow velocity in conventional intake valve is more in pipe side than liner part. Thus, turbulence generated in conventional intake valve was more than that in case of modified intake valve with plate. Thus, using the modified design, the better results are obtained in terms of air fuel mixing.

B. Further Recommendation

- The comparative analysis of two different configurations of intake valve designed for variable compression ratio is performed. The modified valve design can be used for further analysis.
- Reverse pressure, swirl motion, swirl ratio, discharge number etc. parameters can further be decided by analysing the design
- The present analysis may be further extended as; the most of focus in this study is given towards the investigation of air flow, motion for calculating velocity and turbulence energy while considering the valve lift.

C. Limitations

Objective of present investigation is to have study of existing intake valve and modified design of intake valve with thin plate type structure and to evaluate the motion of air flow from it. During the study of these intake valve velocity of air is taken into account and neglecting its effect on other parts of engine. In this study the some of the limitations which are observed are as follow, the scope of the study is only restricted with the variable compression ratio engine and no other types of engines are considered during this study. The study consists of use of modified intake valve design and it has been not evaluated for the engines operating with alternative fuels.

VII. CONCLUSION

The flow of fuel inside of internal combustion engine will be affected largely due to Air flow motion because ultimately the combustion of fuel depends upon the air flow motions. The effect of axial and radial velocity of air flow in intake manifold will ensure good performance of engine. Velocity of air should be more near valve ensuring complete blend of air and fuel charge, velocity of air in liner part is very important. Higher will be the velocity in that region more turbulence will be created. Which will lead to better swirl motion. More turbulence also ensures better air and fuel mixture. The study is conducted for the air flow motion investigation considering six conditions of valve lift from these cases we considered in this research work we can conclude that The air flow velocity in conventional intake valve is more in pipe side than liner part. Thus, turbulence generated in conventional intake valve was more than that in case of modified intake valve with plate. Thus, using the modified design, the better results are obtained in terms of air fuel mixing. The In-cylinder flow velocity of air is observed as low than that around valve, in all six cases of valve lift.

Due to sudden rise in sectional area near engine outlet, the air flow velocity is seen as rapidly increasing. Maximum air flow streamlines tend to move towards bottom of the cylinder. In case of conventional valve, the tendency of air spreading inside of chamber is more than that with modified valve.

REFERENCES

1. A. Raj Kumar, G. Janardhana Raju and K. Hemachandra Reddy, (2016) Comparison of swirl turbulence generating devices in compression ignition engine. Archives of applied science research, 8 (7): 31-40.

2. Arvindkumar K., Adhithiyan, N., Darsak, V. S., and Dinesh, C.D. (2014) Optimization of intake manifold design using fiber reinforced plastic. International journal of scientific and engineering research, 5 (4): 922-925.
3. Aljamali, S., Mahmood, W. M. F. W., Abdullah, S., and Ali, Y. (2014). Comparison of performance and emission of a gasoline engine fueled by gasoline and CNG under various throttle positions. Journal of supplied sciences, 14: 386-390.
4. Angadi, B. M., Malipatil, A. S., Nagathan, V. V., and Kattimani, R. S. (2003) Modelling and Analysis of Intake manifold of a Multi-cylinder SI engine. International Journal of Conceptions on Mechanical and Civil Engineering, 1(1).
5. Y. Yorozu, M. Hirano, K. Oka, and Y. Tagawa, "Electron spectroscopy studies on magneto-optical media and plastic substrate interfaces(Translation Journals style)," *IEEE Transl. J. Magn.Jpn.*, vol. 2

AUTHORS PROFILE

Asst. Prof. Pankaj R Pardeshi,
Working in Dr. D Y Patil institute of Technology Pimpri Pune Mechanical Engineering Department
ME Heat Power, Specialization Energy Audit and Management Thermal Engineering, Energy Engineering, Heat Transfer Having 8 Yrs of Teaching and 2 Yrs of research experience

Dr. K K Dhande, HoD Mechanical Dr D Y Patil Institute of Technology Pimpri Pune ,
PhD. Mechanical,
Specialization Mechanical System Design, Tribology , Design of Machine Element . Having 27 Yrs of Experience in teaching and research filed

Asst Prof. Gorkh P Bhagat
M.E. Heat Power
Specialization: Heat Transfer , Thermal Engineering , IC Engine, RAC, Turbo Machinery , Fluid Mechanics, Energy Engineering Having 10 Yrs of Experience in Teaching and 2 Yrs of Research Experience

Asst Prof. Vikram S Suvarnkar
M.Tech – CAD CAM
Specialization Heat Transfer , Thermal Engineering , IC Engine, Turbo Machinery , Fluid Mechanics, Energy Engineering TOM Mechatronics Having 12 Yrs of Experience in Teaching and 3 Yrs of Research Experience

Asst Prof. Vijaykumar K. Javanjal
M.E. Heat Power
Specialization:, Thermal Engineering , IC Engine, Turbo Machinery , Fluid Mechanics, Energy Engineering TOM Mechatronics Heat Transfer Having 16 Yrs of Experience in Teaching and 3 Yrs of Research Experience