International Journal of Engineering Technology Science and Research
IJETSR
www.ijetsr.com
ISSN 2394 – 3386
Volume 5, Issue 1
January 2018

Study on CFD Analysis of IC Engine Manifolds

Amit M. Patil
Research Scholar, JJT University, Jhunjhunu, Rajasthan, India
Dr. Ghanshyam Shankar Tasgaonkar
JJT University, Jhunjhunu, Rajasthan, India

ABSTRACT

The aim of this paper is to summarize numerical investigation done by various researchers towards IC Engines manifold design modifications. The modifications done by designers are related with improvement of air entry into the intake manifold which will result in efficient combustion of air fuel mixture. During this study numerical performance evaluation of various configurations of geometric modifications of manifolds are reported .Through numerical simulations air velocity its effect on power output, volumetric efficiency, turbulence and effect of swirl are studied. The CFD analysis is performed by using Ansys Fluent software. The mesh grid independency study is also conducted which will aid in improving the accuracy of results.

Keywords (Air, Analysis, CFD, Engine, Simulation)

INTRODUCTION

The computational fluid dynamics is powerful tool in the investigation of flow behavior of different types of fluids. In case of internal combustion engine the air flow is simulated with CFD which helps in the understanding of its effect on engine performance as well as exhaust gas emissions. Today aim of all the engine manufacturers is to reduce exhaust gas emission because air pollution increases with exhaust gases. The simulation results obtained from CFD will acts as a basis for modifications in the engine manifolds for various researchers and designers. During CFD analysis most important aspect is to define boundary conditions. The accuracy of results largely depends upon the correct boundary conditions. This paper consists of data of boundary conditions in the numerical analysis of intake manifold, meshing data and results obtained.

LITERATURE REVIEW

Bandi Ramanjuluet al. [1] Numerical analysis performed in Ansys fluent and result shows that, helicalspiral manifold geometry gives higher velocity. Whereas spiral manifold gives higher Swirl ratio inside the cylinder and turbulent kinetic energy. As far as Volumetric efficiency is considered for the spiral helical combined manifold is 10% higher than that of spiral manifold. Helicalspiral combined manifold creates higher swirl inside the cylinder than spiral manifold. The effect of change in crank angle on swirl ratio is presented by the author. K.M Pandey et al. [2] CFD invetigation is carried out for Poppet type valve. During the numerical analysis various boundary conditions such as mass flow of air at inlet having 72° crank angle which is equal to 0.01319 kg/sec, wall boundary condition is given to cylinder solid surface at temperature of 300°K. pressure outlet is given to piston having pressure of 0:935 bar, injector of the valve is defined as discrete phase surface injection with fuel flow rate of 0.0011 kg/sec. The grid independence study shows that 384876 cells and 82377 nodes are generated which will improve the accuracy of analysis results. The result shows that, the intensity of swirl decreases with respect to stroke length of cylinder. D Raj Kamal et al. [3] CFD analysis was performed at 72°and 123°crank angle for high swirl induction intake valve. The result shows that tangential velocity produced by intake charge is higher than tangential velocity produced by Poppet valve. The computational analysis is performed by using Standard k omega turbulence model. The use of such type of swirl will reduce exhaust gas emissions. Abdul Rahiman et al. [4]. The numerical simulation is carried out with RNG k-ε model due to complexity of the geometry and fluid motion. The CFD simulations consists of analysis intake manifold using dynamic mesh approach, effect of inlet manifold configurations (helical, spiral, helical spiral) on the in cylinder flow, Comparative study of effect of different inlet manifold configurations with straight manifold on swirl ratio, tumble ratio, volumetric efficiency & turbulence in the engine. For computational domain Pressure inlet boundary conditions is used to define the fluid pressure at flow inlets with pressure of 1 atmospheric. Engines walls defined as stationary no slip walls. Reynolds number was considered approximately as 50,000. The result shows that Spiral and helical-spiral manifold creates higher swirl than Normal inlet manifold. Helical manifold gives higher TR& turbulent kinetic energy than Spiral and helical spiral manifold. Volumetric efficiency washigherforhelical manifold. It is suggested that helical-spiral manifold has advantage of average volumetric efficiency, hence need to be used in engines. A.S.Phulpagar et al. [5] simulated air intake system with filter and investigated air flow characteristics for filter geometry and filter media. Unstructured tetrahedral meshing is used. The CFD analysis is performed using Standard (k-ε) turbulence model. The boundary conditions are defined as Inlet Pressure equal to atmospheric pressure, Outlet Mass flow at 6000 rpm is taken as 312 kg/h and Filter Porosity as 0.85. The numerical analysis is carried out by varying mass flow rate based on engine rpm. Pressure drop in intake system was calculated by taking pressure difference between inlet and outlet. Based on results obtained geometry modification is required due to low velocity and flow separation of air. P. D. Solanki et al. [6] During CFD simulation grid independent tests are performed which shows that n umber of nodes varies between 240223 cells to 1309839 cells. Cylinder pressure is applied as Boundary conditions at inlet of the plenum chamber and at the exit of valve, while cylinder wall was specified as adiabatic with no slip condition. Standard K- ε turbulence model is used for the numerical simulation. All the intake manifold configurations are analyzed as steady state with all runners Open, 1st & 3rd runner open and 2nd & 4th runner open. CFD analysis result shows that MODEL 02 is better than other three models because uniform distribution of air with high velocity and less loss of kinetic energy. Ultimately Model 02 has higher swirl which is important for the performance of inlet manifold. Frantisek SEDLACEK et al [7] through number of simulations for resonant pipes varying in length from 18 to 500 mm and intake manifold volume ranges from 4-8dm^3. The length and volume are selected as 292 and 318 mm and 6.15dm^3. Out of number of intake manifold configurations the design chosen as the inlet of the manifold is located in the longitudinal axis of the intake manifold and runners are located in line in the transverse axis which gives better air distribution. The turbulence model used is SST (Shear stress transport; k-ω) with 300000 up to 5 million 3D tetrahedral elements of mesh obtained through mesh independency test. The boundary condition consists of outlets (for the individual cylinders) one applied temperatures, and pressure and for inlet of the intake manifold was applied static pressure and temperature. During optimization process maximum mass flow through the intake manifold was defined as objective function of the geometric optimization and the total volume of the Intake manifold was defined as the constraint of the optimization. Result indicates that all the cylinder are filled uniformly with variation of 2.2 %. Mohd Faisal Hushim et al. [8] studied change in the intake manifold angle on the performance of engine using CFD. The various angular configurations of intake manifold varies from 30°, 60°,90°,120°,150° and 180°.The turbulence model selected as standard K-efor analysis. The various boundary conditions listed are as inlet pressure 101.3 kPa, Outlet pressure 0 Pa, Intake manifold wall taken as No slip wall and smooth wall roughness , Air velocity 15 m/s and Air temperature 300 K. The air flow need to be homogenous which will give better performance output. The result concludes that intake manifold with angle of 180° gives better air motion due to low air resistance than low manifold angle. Swapnil Vilas Nimkarde et al. [9] Numerical simulation is carried out with Reynolds number k − ε turbulence models with associated wall functions. The boundary conditions during CFD analysis are taken as Initial swirl ratio 2, Initial charge temperature 363K, Average Cylinder wall region temperature and region temperature 400 K. The CFD simulation results are validated with experimental results. The result shows that with the Central beads in the manifold along with the concentration of the NO emissions are 60% lower than threaded with Normal type and 66% lower than other intake manifolds. S.K. Sabale et al. [10] the proposed helical inlet port is simulated in CFD for each valve lift and results are correlated with experimental set up. The aim of investigation is to have maximum swirl. Numerical analysis consist of refined volume mesh. The swirl ratio increases with decrease in T/R ratio. Suresh Aadepu et al. [11] CFD analysis is carried out with regular k-e turbulence model. The number of node in first model were 54033 and for second model 55637. The boundary conditions are considered as Inlet of manifold Pressures 0.991& 0.980, Exit of runner to Cylinder 1 mass flow rate, g/s 60.88 &0.45 , Exit of runner to Cylinder 2 mass flow rate, g/s -0.27& 62.32 at crank angle of 448.83 and -83.35 respectively. The result indicates that volumetric efficiency for existing design increases by 7 %. It also suggests that changes in geometry of plenum also improve intake manifold performance. Giovanni Vichia et al. [12] Numerical results has shown that the plenum need to be modified for variable geometry intake system. The authors have simulated four different plenum types viz. Intake line without the plenum, Intake line with the plenum always open, Intake line with the plenum open only when WOT condition is reached and Intake line with the plenum open only whenTVO overcomes the 60% (CL1). The Simulations in stationary and transient conditions have shown that it is better to exclude the plenum from the suction line both at low engine speeds and when the throttle valve opening is less than 60%. A. Raj Kumar et al. [13] CFD analysis is performed using standard K-Epsilon Model with inlet velocity considering as 35m/s and pressure of 100Kpa. The six vane type device produces maximum swirl than other models. As pressure drop in six vane type model is less hence it is suitable for fitted into intake manifold of engine.Rajesh Holkar et al. [14] authors have performed numerical simulation of steady air flow through intake manifold of IC engine.The objective of study is to Uniform distribution of air to all cylinders along with Minimum possible resistance in inlet manifold runners. Numerical simulation is performed using FLUENT and turbulence model used as RNG k-epsilon model. The boundary conditions applied are at outlet zero gauge pressure, wall boundaries have a no slip condition and at inlet standard wall function mass flow rate at inlet is 0.928156 m/s. Meshing gives 315410 tetrahedral cells with grid independency tests. The 3 Dimensional numerical simulation of the air intake region assists for predicting pressure loss affected by the physical configuration. The flow is dependent on the valve lift except upstream of the port bend.Losses in the valve clearance are higher whereas at higher valve lift flow separation becomes complex in nature. Sachin Singla et al. [15] The CFD analysis of these three geometries is performed and results are validated with experimental procedure. The analysis result shows that Model with better design of runners and the curves at the end gives equal velocity at outlet. Velocity variation is observed due to faulty design of plenum, the results are largely affected by Position of the runners with respect to the inlet. Low velocity and high pressure loss are reported in Model 1 due to barrier to air flow. Modified design give same velocity at each outlet of runner as compared to original mode1of intake manifold with increase of 16% in flow velocity at outlet 1, and velocity in other outlets increased by approximately 5% to 7%. Madhusudan Barot et al. [16] Numerical simulation is performed to find out mass flow rate of air with variable valve lift such as 2mm, 4mm and 6mm and with increase in valve lift mass flow rate increases. The intensity of swirl reduces along the stroke length of the engine cylinder.M. A. Jemni et al. [17] The CFD analysis is based on standard k-ε model. The boundary conditions are defined as inlet intake manifold pressure equal to 1.013 bar for the air inlet and 1.5 bar for the propane inlet. The alternative piston speed along the intake stroke is taken as final condition to 3.55 m/s. The meshing result shows that number of nodes are 22632. Numerical results are compared with Experiment tests which show that, the air-fuel ratio and the specific fuel consumption are measured and improved by 7 % and 28 % respectively. Dr. Hiregoudar Yerrennagoudaru et al. [18] CFD simulations are performed by varying valve lift; the total valve lift is 12 mm and divided into three different types. The valve lifts are Low Lift (valve at 4mm downward movement), Medium lift (valve at 8mm downward movement) and High lift (valve at12mm downward movement). The numerical results are compared with experimental results and states that Masking of inlet valves and Fins improves swirl rate and intern brake thermal efficiency of engine. The exhaust gas emissions of other two valve are less in comparison with base valve. The cost of masks and fins are less. Pandey K. M. et al. [19] CFD analysis is done with 72°and 123°crank angle with boundary conditions as inlet boundary condition is assigned as mass flow inlet, Solid surface of the cylinder of the engine is assigned wall boundary condition i.e. no slip condition on the solid surface of the cylinder, Outlet Boundary on the piston of the engine Outlet boundary is assigned the pressure. The result shows that Poppet valve designs better than other tow geometries because it produces highest swirl in intake manifold. As valve lift increases swirl intensity is reduced for all there inlet valve geometries. Piotr Swiatek et al. [20] Numerical simulations are performed on manifold with standard K-e model with ANSYS- AIM code. Boundary conditions are Air mass flow rate of 60 g/s at inlet and temperature as 15°, gauge pressure of -0.1 bar is defined at outlet, Manifold walls are stated as adiabatic wall. After meshing number of elements were obtained as 630000. The flow bench test are conducted on manifold 2 only. The special intake manifold flow test bench measure the mass airflow in each of the manifold outlets and analyze its distribution for multi-cylinder engines. The result also shows that application of a guide vane inside the intake manifold improves the uniform distribution of the airflow between the cylinders. Bayas Jagadishsingh G. et al. [21] Numerical simulations are performed using 1-D CFD code Lotus software. Experimental analysis is performed on intake manifold with runner length of 250mm and diameter of 45mm. Results obtained from analysis as with engine speed rise the length of intake manifold has to be reduced in order to achieve optimum performance of engine. Volumetric efficiency, brake torque & brake power increases as 6%, 17% and 10% for variable intake lengths.Jianmin Xu [22] In CFD analysis tetrahedral meshing is done by software GAMBIT and Three dimensional flow simulation was conducted using software FLUENT. The Standard turbulence model k –e is used , front surface of the inlet end is set to mass flow inlet boundary, outlet face of each manifold is set to pressure outlet boundary and others are set to wall boundary.The result shows that design 3 has good flow characteristics than other two designs of intake manifolds.

CONCLUSION

The numerical simulations performed for the analysis of engine manifolds are reported in this paper. The numerical simulations are performed in order to study the effect of geometric changes in the design of manifold of engine. The geometric modification consists of length, diameter and angle variations. The standard k-e turbulence model is used by all the authors for numerical simulations. The mesh independency study give maximum number of elements in meshing, which results in accurate results. The results obtained from numerical investigations are correlated with experimental results by some of authors.

REFERENCES

[1] Ramanjulu, B., Adissu Fulli, D., & Bekele, A. E. (2015) “Performance Analysis of IC Engine Based on Swirl Induction by Using CFD”, International Journal of Advanced Research in Science, Engineering and Technology, 2 (5), 622-627.
[2] Pandey, K. M., & Roy, B. (2012). CFD analysis of intake valve for port petrol injection SI engine. Global Journal of Research in Engineering, 12(5-A).
[3] M D Raj Kamal,S.Kaliappan, S.Socrates, G.Jagadeesh Babu (2017). CFD Analysis of Single Cylinder IC Engine Inlet Swirl Valve Computers & fluids. International Journal of Latest Engineering Research and Applications, 2(8), 34-36.
[4] Rahiman, A., RK, A. R., AD, M. S., & Ramis, M. K. (2014). CFD Analysis of Flow Field Development in a Direct Injection Diesel Engine with Different Manifolds. American Journal of Fluid Dynamics, 4(3), 102-113.
[5] Phulpagar, A. S., & Gohel, N. S. CFD Analysis of Air Intake System. International Journal on Theoretical and Applied Research in Mechanical Engineering, 4.
[6] Solanki, P. D. (2015). Simulation Studies on Different Intake Manifold Designs for Performance Improvement of Four Stroke Four Cylinder Diesel Engine. Mechanical Engineering Department, Student of Master of Engg.(IC Engine and Automobile) SVMIT, Bharuch, Gujurat, India.
[7] Sedlacek, F., & Skovajsa, M. (2016). Optimization of an Intake System using CFD Numerical Simulation. Proceedings in Manufacturing Systems, 11(2), 71.
[8] Hushim, M. F., Alimin, A. J., Razali, M. A., Mohammed, A. N., Sapit, A., & Mendez Carvajal, J. C. (2015). Air flow behaviour on different intake manifold angles for small 4-stroke PFI retrofit kit system.
[9] Swapnil Vilas Nimkarde, Anant S. Bombatkar, (2017). Investigation and Analysis on the Effect of Homogeneity Enhancement of Charge for S.I. Engine, 5(1), 457-466.
[10] Sabale, S. K., & Sanap, S. B. (2013). Design and Analysis of Intake Port of Diesel Engine for Target Value of Swirl. American Journal of Mechanical Engineering, 1(5), 138-142.

[11] Aadepu, S., Prasanth, I. S. N. V. R., & Naik, J. M. (2014). Design of intake manifold of IC engines with improved volumetric efficiency. Internatıonal Journal & Magazıne of Engıneerıng, Technology, Management and Research, 1(6), 26-33.
[12] Vichi, G., Romani, L., Ferrari, L., & Ferrara, G. (2015). Development of an engine variable geometry intake system for a Formula SAE application. Energy Procedia, 81, 930-941.
[13] 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.
[14] Holkar, R., Sule-Patil, Y. N., Pise, S. M., Godase, Y. A., & Jagadale, V. S. (2015). Numerical simulation of steady flow through engine intake system using CFD. Journal of Mechanical and Civil Engineering, 12(1), 30-45.
[15] Singla, S., Sharma, M. S., & Gangacharyulu, D. Study of Design Improvement of Intake Manifold of Internal Combustion Engine. International Journal of Engineering Technology, Management and Applied Sciences ISSN, 2349-4476.
[16] Madhusudan Barot, Prof.Abhishek Shah, Prof. Mit Patel (2017) CFD Analysis of Single Cylinder Four Stroke Gas Fueled Engine for Prediction of Air Flow Rate during Suction Stroke. International Journal of Engineering Development And Research,5(2), 1136-1140.
[17] Jemni, M. A., Kantchev, G., & Abid, M. S. (2012). Intake manifold design effect on air fuel mixing and flow for an LPG heavy duty engine. International Journal of Energy and Environment, 3(1), 61-72.
[18] Dr. Hiregoudar Yerrennagoudaru, Shiva Prasad Desai (2014). Effect of Inlet Air Swirl on Four Stroke Single Cylinder Diesel Engine Peformance.International Journal of Recent Development in Engineering and Technology. 2(6),95-103.
[19] Pandey, K. M., & Bidesh, R. (2012). Intake Valve Design for High Level Swirl induction in Carbureted Spark Ignition Engine. Journal of Environmental Research and Development, 7(1A).
[20] Swiatek, P., Fuc, P., Cebula, A., & Kowalczyk, S. (2017). Flow bench testing of prototype intake manifolds for ultralight aircraft engine. In MATEC Web of Conferences (Vol. 118, p. 00008). EDP Sciences.
[21] Bayas Jagadishsingh, G., & Jadhav, N. P. (2016). Effect of Variable Length Intake Manifold on Performance of IC Engine. International Journal of Current Engineering and Technology E-ISSN, 2277-4106.
[22] Xu, J. (2017, October). Flow analysis of engine intake manifold based on computational fluid dynamics. In Journal of Physics: Conference Series (Vol. 916, No. 1, p. 012043). IOP Publishing.