International Journal on Recent and Innovation Trends in Computing and Communication ISSN: 2321-8169
Volume: 6 Issue: 4 28 - 34

Gas Flow Analysis inside Combustion Chamber of Two Stroke SI Engine

Saurabh Meshram, Satyam Nimbulkar, Suhel Dhargave, Saurabh Jambhale, Shubham Patil, Sachin Kasbe (Students)

Mr. Shailesh Dhomne
Assistant Professor,
Dept of Mech. Engg,
Nagpur.

Abstract:- Computational Fluid Dynamics has become an essential tool in the design and development process of IC-engines. Due to increasing computer hardware performances and steady decreasing costs it is to expect that simulation technology becomes an unavoidable tool for the analysis of many technical problems. In present study the flow analysis of the charge in the cylinder of 2 stroke SI Engine using CFD ANSYS fluent is carried out. The variation of pressure, velocity, temperature of charge during its flow from inlet to outlet is studied. The effect of thermal barrier coating (TBC) on the piston crown is analysed. This study may help to optimize the engine performance.

*****

INTRODUCTION

The conventional two-stroke spark ignition engine is technically a very simple power plant. With the exception of the uniflow scavenged engine, it does not use any valves in its operation. The intake and exhaust processes are controlled by the reciprocating piston itself. After combustion, the piston moves downward due to rapid rise in pressure and temperature above the piston. Below the piston, the opened inlet port induces air from the atmosphere into the crankcase due to the increasing volume of the crankcase lowering the pressure below the atmospheric value. In a simple design of two-stroke engine, a carburettor is placed in the inlet port for fuel induction. In the downward stroke, the piston uncovers the exhaust port releasing a pulse of hot, high-pressure exhaust gas from the cylinder to the atmosphere through the exhaust duct. This is generally referred to as blow down process.

At this stage, the compression of fresh charge in the crank case below the piston (crankcase compression) takes place. As the piston continues to move downward, it opens the scavenge or transfer port which connects the cylinder directly to the crankcase. When the crankcase pressure exceeds the cylinder pressure, fresh charge enters into the cylinder through the transfer port and forces combustion products out of the cylinder. This process is called scavenge process. Due to the fact that the transfer and the exhaust ports are open, the incoming fresh charge to the cylinder through the transfer port can exit directly out of the exhaust port during this period.

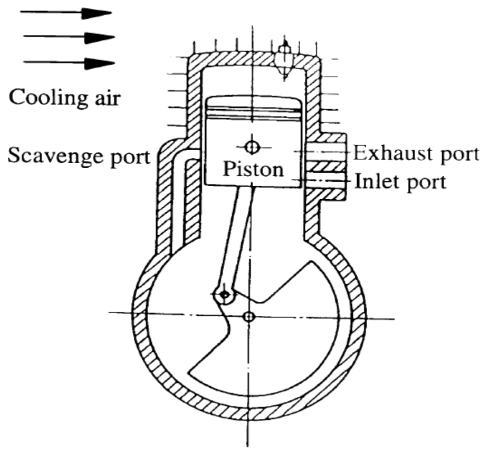

Fig. 1 Two stoke cycle engine

Computational Fluid Dynamics (CFD) has become an essential tool in the design and development process of engineering devices. In the beginning, it was mostly limited to high technology engineering fields, e.g. aeronautics and astronautics, but in the last decades it has become a widely used tool for the analysis of complex technical problems in many modern engineering applications. Computer modelling allows virtual prototyping of a technical system in order to evaluate the performance of a new design and/or new technologies. This technique is used in the industry in order to reduce the development and retrofitting time and therewith the product costs.

The considered problems may involve complicated geometry and physically complex phenomena such as turbulence, phase change, chemical reactions, etc. In the last decades, CFD has also become a state of the art tool for the development of internal combustion engines (IC-engines). It offers the successful assessment of new technologies, e.g. new fuel preparation methods, new combustion concepts and/or alternative fuels.

Currently, different CFD software packages are in use, such as ANSYS. FLUENT, STAR-CD, KIVA, CONVERGE CFD, OPENFOAM etc. Some of them are especially developed for the simulation of IC-engines and therefore provide all required models and mesh tools. Other software packages like ANSYS FLUENT, can be applied to a wide array of industrial applications and are developed with a general purpose.

REVIEW OF LITERATURE

Through various research papers and journal papers literature survey has been conducted, through which it observe that most of the work is done on two stroke SI engine also the gas flow analysis in combustion chamber is require to enhance the performance of two stroke SI engine.

The in-cylinder flow field analysis in a 2S engine is studied in this paper by using CFD. The main parameters which are consider are Engine speed and Compression ratio. The experiment is conducted at various speed and then velocity vector fields obtained are analyzed to understand the in-cylinder flow behaviour .The CFD codes are used for carried out the study in this area. Then CFD results are compared with the actual experimental results. The boundary conditions apply are pressure with crank angle and cycle avg. pressure data from MATLAB By considering all the results they conclude that the port orientation, rather than port areas had a greater influence on the in-cylinder flow parameters.[3]

When the air fuel mixture is burnt in the combustion chamber the Heat of hot gasses is from top surface of piston is conducted to the other side of piston and then to the engine for that heat transfer to an engine piston is calculated. For this purpose the software KIVA code are used. Here numerical values are compare with the experimental values and give the conclusion about the analysis. Here the geometry is created in solidworks and then NASTRAN is used for the thermal analysis and PATRAN is used for mainly meshing which is further used for analysis.

In this the different boundary conditions are apply i.e thermal boundary conditions and combustion side boundary conditions and conclude that applying the transient boundary condition is very time consuming and does not affect the results of piston thermal analysis within engineering approximations.[16]

The investigation of the combustion stability of spark ignition engine by using numerical and experiment analysis by ENZO GALLONI and characterizing the engine reproduced by CFD analysis and performed by RANS and calculated various parameters like mean flow , mean laminar flame speed and the mean turbulence intensity .[15]

In I.C engines with different case as like velocity, temperature and pressure boundary conditions in experimental setup and also concerned with studies aimed at understanding type of flows in cylinder. The geometric model and mesh is developed by using ANSYS with the help of CFD and further analysis is done by the ANSYS.[19].

The study is about the effect of piston configurations on in-cylinder flow. The results from the modelling and CFD simulation using FLUENT software in this paper are shown in term of graphs for the simulation results for pressure distribution, temperature distribution and Velocity.[21]

The purpose of this paper is to study the Computational Fluid Dynamics (CFD) modelling of the combustion process using detailed chemistry in a spark-ignited (SI) optical access engine operated at part load using gasoline and ethanol as fuels. Simulation results are compared against experimental optical and indicating data. The study presents a simulation work aimed at predicting the combustion behaviour in an optical access SI engine fuelled with gasoline and ethanol. Ethanol and gasoline combustion speed, at various air-fuel ratios are well described by the model.[8]

The flow characteristics inside the engine cylinder equipped with different piston configurations is carried out in this study. For this, complete calculations of the intake and compression strokes were performed under realistic operating conditions and the ensemble-averaged velocity and turbulence flow fields obtained in each combustion chamber analyzed in detail. The results confirmed that the piston geometry had little influence on the in-cylinder flow during the intake stroke and the first part of the compression stroke. However, the bowl shape plays a significant role near TDC and in the early stage of the expansion stroke by controlling both the ensemble-averaged mean and the turbulence velocity fields.[18]

The Heat transfer from the hot gases to the wall in exhaust systems of high-performance two-stroke engines is underestimated using steady state with fully developed flow empirical correlations. This fact is detected when comparing measured and modelled pressure pulses in different positions in the exhaust system. This can be explained taking into account that classical expressions have been validated for fully developed flows, a situation that is far from the flow behaviour in reciprocating internal combustion engines. The heat transfer in the exhaust system of high-performance two-stroke engines was investigated. Different models from the literature, which take into account the entrance length and the flow fluctuations.[6]

3D Modelling of combustion Chamber

ISSN: 2321-8169
28 - 34

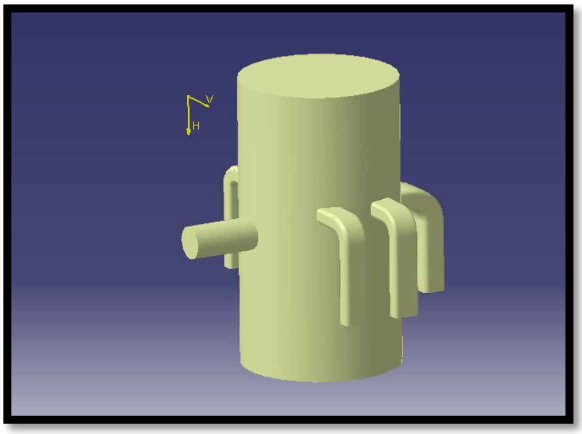

Fig.2 Combustion chamber

The model of combustion chamber is created in CATIA (design software) by using the all standard dimensions of two stroke SI Engine.

In this model we assume that the piston is at the bottom so the body of combustion chamber is fully closed. Since the piston is at the bottom dead center we can see the gas in maximum volume of the combustion chamber. The above combustion chamber model is created in surface design of CATIA software having only surfaces hence having no thickness because we will want to analysis gas flow inside combustion chamber. According to the problem the modelling is done by using the CATIA.

The transfer ports and inlet port of the model of combustion chamber are closed because in analysis software the inlet boundary conditions and outlet boundary conditions are given at the inlet port and outlet port . according to this the modelling is done in CATIA.

CFD Analysis by ANSYS fluent

ANSYS is used for ovelall analysis of the the combustion chamber of two stroke SI Engine. The ANSYS consist of various solver like Fluent, Transient Thermal, etc. In this gas flow analysis we use the fluent.

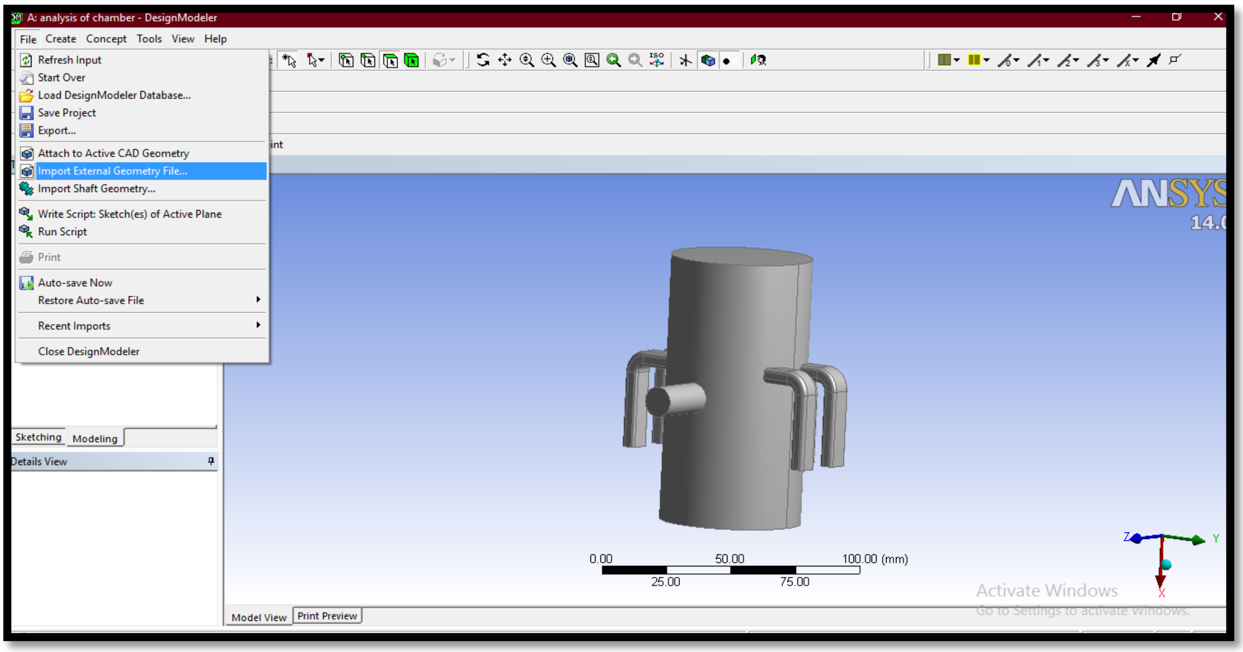

Fig.3 Geometry import in ANSYS

First process of any analysis is to import the geometry which is drawn in any software in the appropriate format. The combustion chamber geometry is imported in the ANSYS Workbench in igs fromat which is used for further analysis.

Mesh Generation Process

Generally more the no. of elements, more the accuracy of the results but it will converge at some point after that point increase in no of elements does not affect the results. In the meshing of combustion chamber of two stroke SI engine the no. of tringular nodes are generated which are differetiate the geometry into small parts for smooth results.

Problem set up

For finding the gas flow analysis first we have to give proper problem set up for the meshed geometry. In problem set up we have to select proper material, cell zone cconditons, Boundary conditons etc.

Following processes gives the explaination about problem set up for gas flow analysis.

1. General	**Solver**	**Type**	**Pressure based**
		Velocity formulation	**Absolute**
		Flow	**Transient**
		Gravity	**ON**
2. Material	**Air**	**Properties** **a) Density = 1.225 kg/m** **b) specific heat = 1006.43 J/kg-k** **c) Thermal conductivity = 0.0242 W/m-s** **d) Viscosity = 1.789e-05**	
3. Boundary Conditions	**Zone**	**Inlet**	**a) velocity magnitude = 12 m/s** **b) pressure = 1 Bar** **c) Temperature = 300k**
		Interior	**a) Temperature= 500K**
		Outlet	**a) Temperature= 500K**

Analysis of piston

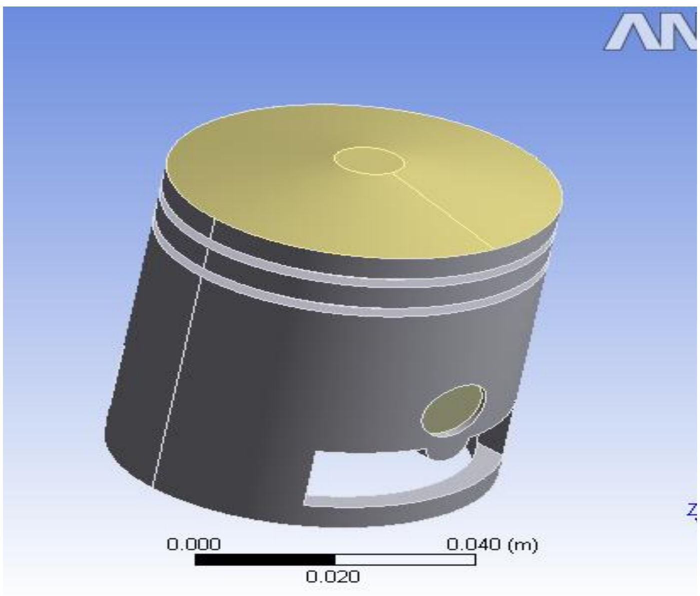

Fig. 4 Piston imported in ANSYS

In the given study we have to analyse the piston with and without thermal barrier coating (TBC). For that the model of piston is created in CATIA and then imported into the ANSYS Fluent. Mesh generation process is also apply and the give the setup for thermal analysis.

In this case we give the temperature boundary condition at the top of the piston and we have to find the heat conduction through piston in both coated and without coated condition.

Boundary condition : 400K Temperature at the top of the piston

RESULTS AND DISCUSSIONS

The variation of pressure, velocity and temperature in the combustion chamber is obtained through CFD. The gas flow characteristics also studied with contours the at various planes in the combustion chamber as explained below:

1) Pressure contours

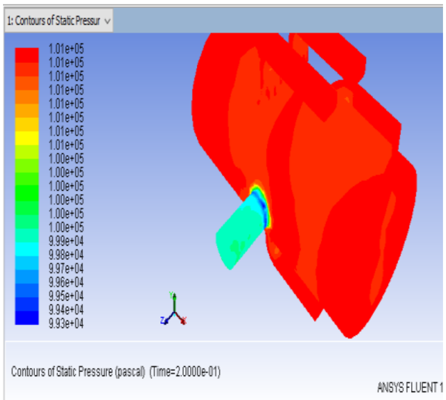

Fig.5(a) pressure variation

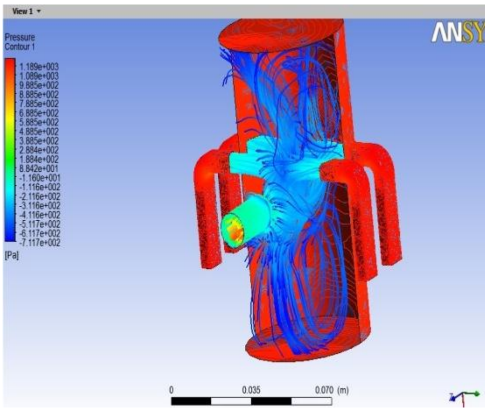

Fig. 5(b) Half cut section of pressure contour

Fig.5(a) shows the pressure at outer body of combustion chamber is maximum. This pressure is decreases at the outlet port. The maximum pressure from given boundary conditions is 1.01 Bar which is at the outer body and minimum pressure is at the outlet port that is 0.993 Bar. Fig.5(b) shows the pressure variation of gas which is flowing inside the combustion chamber.

b) Velocity contour

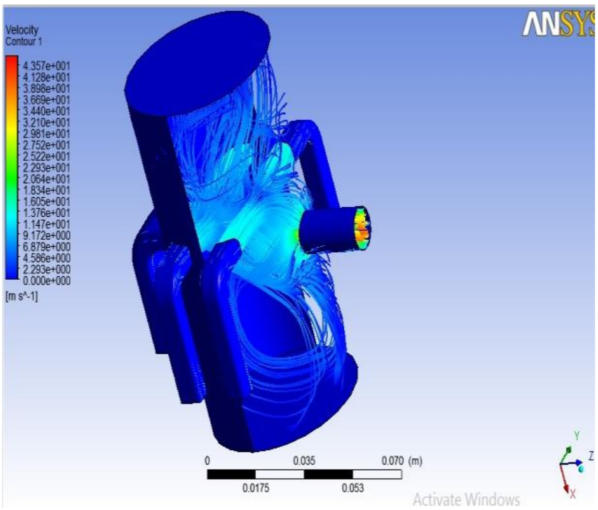

Fig.6(a) Velocity contour

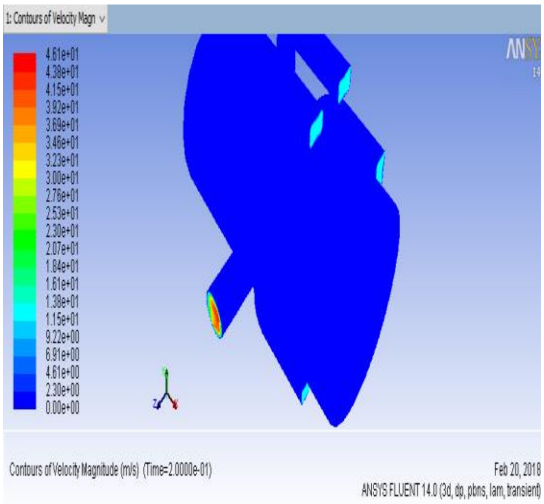

Fig.6(b) Half cut section

By giving the velocity inlet boundary condition at inlet that is 12m/s velocity at inlet port how it is increases upto around 46m/s at the outlet of the combustion chamber is shown by above two figures. For finding the velocity of flow inside combustion chamber we have to cut the half section of combustion chamber which is shown in figure 6(b).

c) Temperature contour

We know that if temperature of gasses are increases inside the combustion chamber then the velocity of gasses is also increases at the outlet port which is shown in figure.

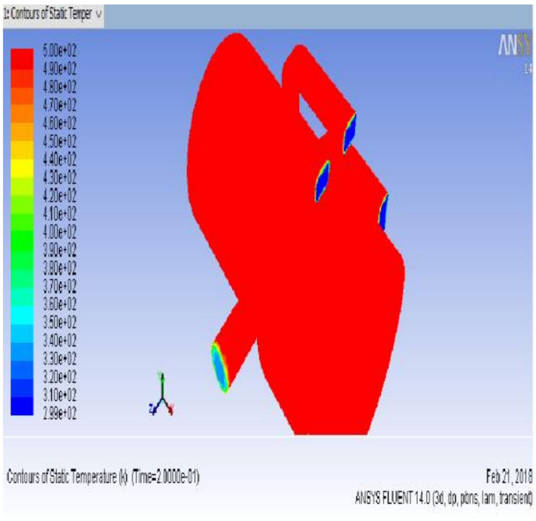

Fig.7(a) Temperature contour

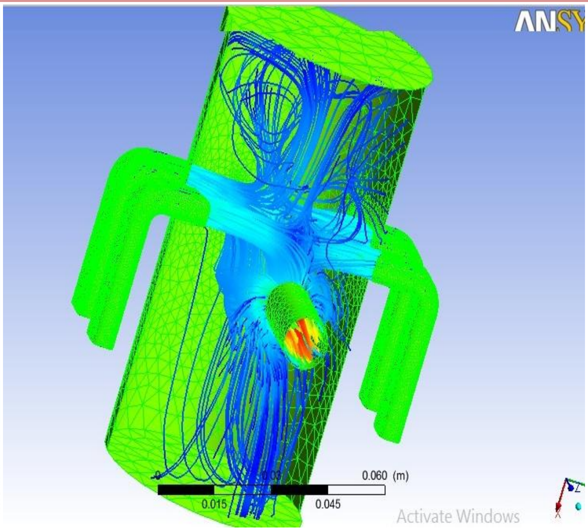

Fig.7(b). Half cut section

The above figure shows the temperature at the inlet port and outlet port. The temperature at the inlet port is given as 300k as a boundary condition. Figure shows the temperature of flow throughout the combustion chamber. The colour shows the different temperature at inlet port and outlet port.

Analysis of piston

The piston of two stroke engine with coating of ceramic material and without coating are analysed and find the heat conduction through the piston.

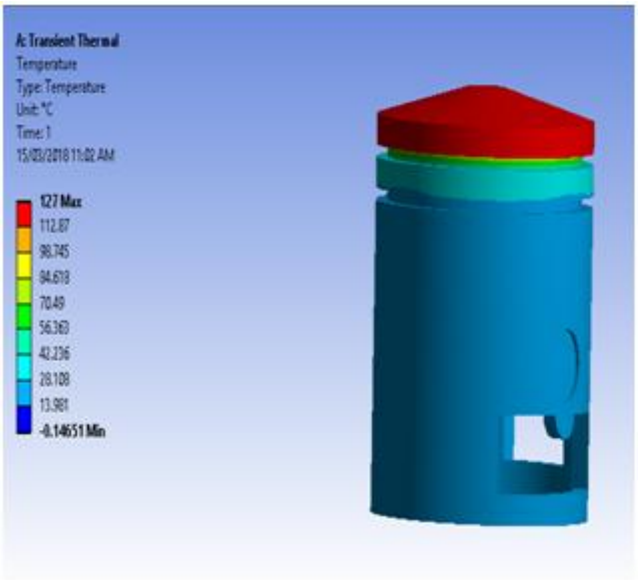

Fig.8(a) For without coated (TBC) piston

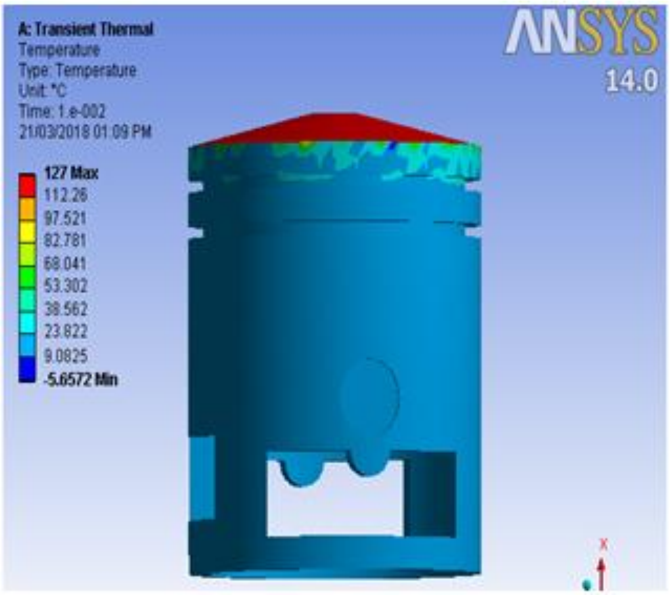

fig.8(b) For TBC Coated piston

For analysis of this piston the initial temperature is taken as 400K t the top of the piston and found that the maximum heat transfer is takes place.

Fig.8(a) shows that maximum heat conduction takes place through without coated piston. The different colours at the top of the piston states that the heat conduction is takes place toward the bottom of the piston. The graph shows the range of the temperature which is reducing from top to bottom.

Due to coating of TBC material the heat transfer is reduced which shows by the given fig.8(b) in which clearly seen that very negligible amount of heat is transfer through piston hence minimum heat conduction is takes place through coated piston.

CONCLUSION

In the present study the gas flow analysis is done inside the combustion chamber. However the analysis of the flow and thermal characteristic of the charge entering and leaving the combustion chamber is analyzed. The temperature, pressure and velocity of flow throughout the combustion chamber and variation in it is analyzed in this study.

The thermal conductivity with and without TBC coated piston is also analysed. It is observed that the heat accumulation in the combustion chamber is increased in case of piston crown coated with TBC material. This improves the combustion efficiency in CC and there by helps in reducing exhaust emissions.

Based on the above analysis it is clear that the overall performance of the IC engine increases by applying the TBC coating in the piston crown and also reduces the exhaust emission.

REFERENCE

[1] Yuh-Yih Wu , Bo-Chiuan Chen, Feng-Chi Hsieh , "Heat transfer model for small-scale air-cooled spark-ignition four-stroke engines " , International Journal of Heat and Mass Transfer 49 (2006) 3895–3905.

[2] Tommaso Savioli , " CFD Analysis of 2-Stroke Engines" , Energy Procedia 81 (2015) 723 – 731.

[3] Addepalli S. Krishna , J. M. Mallikarjuna , Davinder Kumar , "Effect of engine parameters on in-cylinder flows in a two-stroke gasoline direct injection engine" ,Applied Energy 176 (2016) 282–294.

[4] 4.M.Abdul Rahman , "Performance Analysis of Opposed Piston Engine Using Computational Fluid Dynamics" ,International Journal of Research in Advent Technology, Vol.2, No.11, November2014 E-ISSN: 2321-9637

[5] Guven Gonca , "Influences of different fuel kinds and engine design parameters on the performance characteristics and NO formation of a spark ignition (SI) Engine", Applied Thermal Engineering 127 (2017) 194–202.

[6] José Manuel Luján, Héctor Climent, Pablo Olmeda, Víctor Daniel Jiménez, "Heat transfer modeling in exhaust systems of high-performance two-stroke engines", Applied Thermal Engineering 69 (2014) 96e104.

[7] Zhipeng Yuan a, Jingping Liu a, Jianqin Fu a,b, Qi Liu a, Shuqian Wang a, Yan Xia a "Quantitative analysis on the thermodynamics processes of gasoline engine and correction of the control equations for heat-work conversion Efficiency", Energy Conversion and Management 132 (2017) 388–399.

[8] Michele Battistonia, Francesco Mariania, Francesco Risia, Claudio Poggiania Combustion "CFD modelingof a spark ignited optical access engine fueled with gasoline and ethanol",Energy Procedia 82 (2015) 424 – 431.

[9] Vivien Rossbach a, Jonathan Utziga,b, Rodrigo Koerich Decker a, Dirceu Noriler a, Henry França Meier "Numerical gas-solid flow analysis of ring-baffled risers", Powder Technology 297 (2016) 320–329.

[10] K.M Pandey& Bidesh Roy "CFD Analysis of Intake Valve for Port Petrol Injection SI Engine", Global Journal of Researches in Engineering Mechanical and Mechanics Engineering Volume 12 Issue 5 Version 1.0 .

[11] Patil Vijayendra Maharu, Aashish Agrawal, "OPTIMIZATION OF TIME STEP AND CFD STUDY OF COMBUSTION IN DI DIESEL ENGINE", International Journal of Research in Engineering and Technology eISSN: 2319-1163 | pISSN: 2321-7308 .

[12] Maneesh Kumar Dubey and Ravindra Randa, "EXPERIMENTAL PERFORMANCE ANALYSIS OF SINGLE CYLINDER TWO STROKE PETROL ENGINE USING GASOLINE AND LPG" , Int. J. Mech. Eng. & Rob. Res. 2014

[13] M. Cerit,V. Ayhan , A. Parlak , H. Yasar "Thermal analysis of a partially ceramic coated piston: Effect on cold start HC emission in a spark ignition engine", Applied Thermal Engineering 31 (2011) 336e341.

[14] Muhammet Cerit, "Thermo mechanical analysis of a partially ceramic coated piston used in an SI engine", Surface & Coatings Technology 205 (2011) 3499–3505

[15] Enzo Galloni, "Analyses about parameters that affect cyclic variation in a spark ignition engine" , Applied Thermal Engineering 29 (2009) 1131–1137

[16] V.Esfahanian, A.Javaheri, M.Ghaffarpour, "Thermal analysis of an SI engine piston using different combustion boundary condition treatments", Applied Thermal Engineering 26 (2006) 277–287

[17] Yuh-Yih Wu, Bo-Chiuan Chen, Feng-Chi Hsieh, "Heat transfer model for small-scale air-cooled spark-ignition four-stroke engines", International Journal of Heat and Mass Transfer 49 (2006) 3895–3905

[18] F. Payri , J. Benajes, X. Margot, A.Gil , "CFD modeling of the in-cylinder flow in direct-injection Diesel engines", Computers & Fluids 33 (2004) 995–1021

[19] Syed Saleem Pasha, Mohammed Imran, "Compitational Fluid Flow Dynamic Analysis on I.C Engine using ANSYS" , International Journal of Engineering Research & Technology Vol. 4 Issue 05, May-2015

[20] C.S. Sharma, T.N.C. Anand and R.V. Ravikrishna, "A methodology for analysis of diesel engine in-cylinder flow and combustion", Progress in computational fluid dynamic 10(3): 157-167 2010.

[21] H.Sushma and Jagadeesha.K.B , "CFD modeling of the in-cylinder flow in Direct-injection Diesel engine" , International Journal of Scientific and Research Publications, Volume 3, Issue 12, December 2013 1 ISSN 2250-3153.