International Journal of Automotive Engineering, Vol. 10, No. 1, (2020), 3167-3177

Iran University of Science and Technology

International Journal of Automotive Engineering

Journal Homepage: ijae.iust.ac.ir

ISSN: 2008-9899

The study of entrance port shape effect on current rotation rate and observation of combustion behavioral changes in sparking ignition engine with natural gas

Yasin Babajanpour [1*], Davood Domori Ganji [2], Saber Gholipour [3]

1 Sama technical and vocational training college, Islamic Azad University, Babol Branch, Babol, Iran

2 Mechanical Engineering Department, Babol Noshirvani University of Technology, P.O. Box 484, Babol ,Iran

3 Department of Mechanical Engineering, Sari Branch, Islamic Azad University, Sari, Iran

ARTICLE INFO

Article history:
Received : 30 Oct 2018
Accepted: 14 Feb 2020
Published: 1 March 2020

Keywords:
gas engine
output power
pollution
rotation coefficient
air entrance port

A B S T R A C T

Use of natural gas has been proposed as one of the solutions to reduce fossil fuel consumption such as petrol and gasoline, which emit more pollutants. In this regard, more attention has been directed toward use of natural gas due to its high calorific value and low pollution. This paper studies the effect of different fluid rotation coefficients in parallel form with a surface of a piston bowl (Swirl). And, it attempts to explore the changing effects of this indicator on power and major pollutants of sparking ignition gas engines. Three-dimensional computational fluid dynamics are employed to simulate the procedure. Open-cycle engine, the moment between air-intake-valve opens and the exhaust-valve opens, is simulated through applying combustion equations of turbulence and emissions. First, the results are validated based on experimental data. Then, an analysis of different rotation coefficients is used to compare the temperature and pressure inside the cylinder, productivity, and the amount of generated pollution. The results demonstrate that changing the shape of entrance port, which leads to concomitant change in the fluid rotation rate in the chamber, causes a slight change in the output power. But, the change has a significant impact on the production of pollutants.

*Yasin babajanpour
Email Address: yasin_babajanpour@yahoo.com
https://doi.org/10.22068/ase.2020.482

1. Introduction

Today, gas engines are widely use as the main source of generating work and heat with many industrial uses due to their low pollutions. Most of these engines have been converted from diesel mode without having optimized combustion system. They need to be changed in their shape and technology in order to be optimized.

Generally, research and experiments have increased to explore procedures for improving gas engines in recent years. These researches applied different methods to optimize these engines. They have used methods such as reducing of fuel consumption, reducing of pollutions, improving efficiency, and increasing in especial power. Overall, this study could be categorized into three domains of experiments, simulations of thermodynamic and fluid dynamic simulations. Inexorable progress of integrating computer into conducting research brings forth the widespread use of three-dimensional fluid dynamic analysis in research procedures. The effect of different forms of entrance port in the rotation coefficient is studied by Kawashima (1998) [1]. More precisely, he conducted a numerical study on different shapes of entrance valve positions in the four-valve engine with the use of high pressure direct injection. He found out that change in the position of valves should be based on desired swirl in terms of efficiency and reducing pollution in new diesel engines. Kern et al [2] used a swirl meter machine to study swirl in a four-valve diesel engine to detect the order of entrance valves (in a line order or in an inline order). They discovered that the entrance valve in a line order had more vortex effect than the entrance valve in an inline order. Also, they put stress on the role of swirl in reduction of smoke.

In 2011, Prasad et al [3] carried out an empirical study to investigate the effect of rotation rate on the engine’s operation and pollution. The result of their study demonstrated that in special modes of rotation, due to better mixture of fuel and air, rate pollution and fuel consumption were reduced and thermal efficiency was increased.

In 2012, Kumar and Rao [4] achieved different ratios of rotation rate through change of piston surface shape on an experimental basis. These ratios showed that they had less nitrogen oxide and hydrocarbon pollution in an optimized mode while thermal efficiency slightly increased.

The impact of recirculation of air in combustion and emissions in a diesel engine was examined by Prathibha and Prasanthi [5] in a laboratory in 2013. They studied the effect of air recirculation in the cylinder on the function and emissions of cylinders of an engine, with direct injection of four-stroke engines, based on the volume. Grooves were carved out on the piston crown to generate rotation. In this regard, three different configurations from piston with grooves of 6, 9, and 12 were used to tighten rotation and to better mix fuel with air, which had an effect on function and emissions. The experiments tested three pistons based on parameters such as thermal efficiency, specific fuel consumption, delayed ignition, smoke density and emissions of hydrocarbons and carbon monoxide. The result showed that piston with 9 grooves operated better than other pistons in all tested parameters. In a 3D numerical study Jafari and Ganji [6] studied influence of number of nozzle hole on performance and Emission in dual fuel engine in 2013. It is concluded that changing number of nuzzles has a little effect on NOx and CO emission formation in dual fuel engine. Biglarian et al [7] investigated a strategic mild hybrid technology for reducing pollution and optimization of fuel sources in 2014. It is revealed that with an increase in speed and acceleration in a driving cycle, fuel consumption is increased. Environmental impact assessment of replacing conventional taxis with hybrid electric vehicles in Tehran, Iran is studied by Ahmadabadi et al [8] in 2019. It is demonstrated that changing current taxis with hybrid vehicles can satisfy both goals if certain taxes levies on pollution which is generated during different stages and also the price of gasoline and CNG get double.

This study explores the effect of different coefficients of fluid rotation in parallel form with the surface of the piston bowl (Swirl) and attempts to explore the changing effects of this

[DOI: 10.22068/ase.2020.482]

indicator on power and major pollutants of sparking ignition gas engines. AVL FIRE three-dimensional software, which is powerful software for three-dimensional simulation of fluid flow, mixed combination, combined pollution and combustion in sparking ignition engine, is used to analyze the engine behavior. Accordingly, seven different rotation coefficients are considered to be simulated with the software. The achieved results compared with each other.

1.1. Governing equations

In general, current inside the cylinder in internal -combustion engines are three-dimensional, unstable and mixed. For detail analysis of velocity, pressure, and intensity of turbulent flow, it is essential to solve current governing equations which consist of conservation equations of mass, momentum, energy and turbulence. Since these equations are non-linear, coupled, and inter-dependent and finding analytical solution for them is complicated, due to the complexity of the boundary conditions inside the cylinder and boundary changes by the piston movement, numerical approaches were used to find their solutions. The governing equations of Newtonian fluids are as follow [9, 10]:

The equation of conservation of mass:

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho \tilde{u}_i}{\partial x_i} = 0 \quad (1)$$

The equation of conservation of momentum:

$$\frac{\partial \rho \tilde{u}_i}{\partial t} + \frac{\partial \rho \tilde{u}_j \tilde{u}_i}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\mu \frac{\partial \tilde{u}_i}{\partial x_j}\right) - \frac{\partial \tilde{p}}{\partial x_i} + \rho g_i + \tilde{s}_{ui} \quad (2)$$

The equation of energy conservation:

$$\frac{\partial \rho c_p \tilde{T}}{\partial t} + \frac{\partial \rho c_p \tilde{u}_j \tilde{T}}{\partial x_j} = \frac{\partial}{\partial x_j}\left(k \frac{\partial \tilde{T}}{\partial x_j}\right) + \tilde{s}_t \quad (3)$$

Combustion simulations in Fire are able to calculate transferring manner and mixing phenomenon, as well as to simulate combustion in internal combustion engines under premixed, partly premixed and non-premixed conditions. ECFM (E stands for extended) is used in current study. This model is completely coupled with a spray model and is able to model a layer combustion which consists of EGR (Exhaust gas return) effect with nitrous oxide (which defines combustion in sparking ignition engines). ECFM has all the features of the standard CFM model and modified MCFM model.

2. Simulation and networking

2.1. Modeling runner

In addition to the base case figure 1 two other models have been used to analyze the entrance runner in the target gas engine. In these two cases, the implemented change in the upper air on the runner will lead to increase and decrease in rotation coefficient which has been generated by certain difference in pressure between entrance and exit current in the chamber. Paddle Wheel test pattern is used to calculate the runner's rotation coefficients.

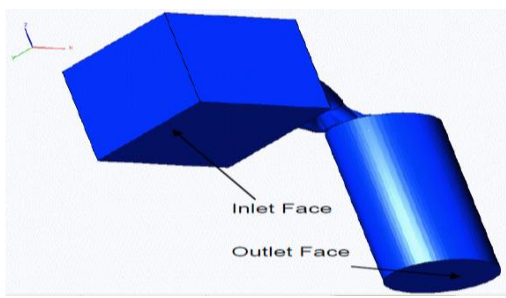


Figure 1: Three-dimensional pattern to create the boundary conditions before meshing

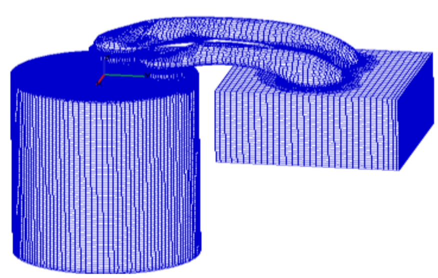


Figure 2: Three-dimensional pattern to create the boundary conditions after meshing

In the base case, calculated coefficient has the amount of 0/472 with the approved experimental simulation.

[DOI: 10.22068/ase.2020.482]

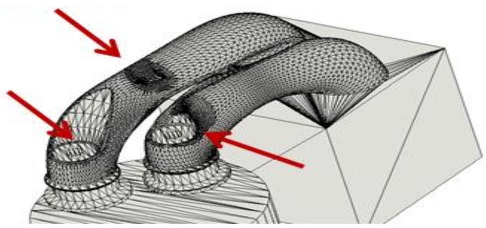


Figure 3: entrance port in base en gine with 0/472 rotation coefficient

Pro Engineering program is used to design runners. After final simulation, Abaqus software was used to mesh it in a triangle format. In this application, the total number of created mesh for the base case is about 8000 cells. This amount was used with a minor percentage difference for the two other cases.

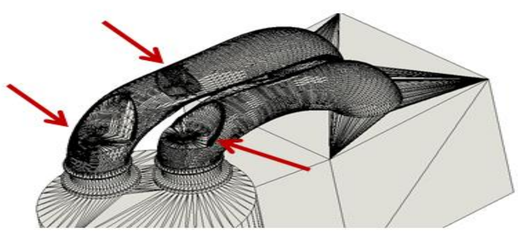


Figure 4: Reduce the rate of rotation (rotation coefficient 0/262)

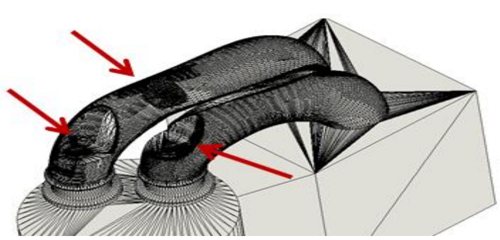


Figure 5: Reduce the rate of rotation (rotation coefficient 0/681)

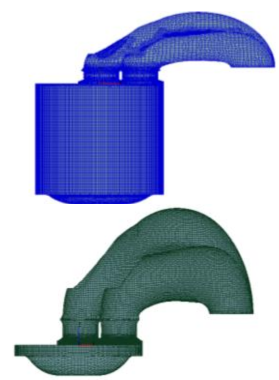
2.3. Combustion modeling

To create three-dimensional models, first, Pro-Engineering software creates three-dimensional model of the piston at the pause point of TCD. At the next stage, the generated geometry is meshed with the use of ABAQUS software (s shown in figure 6.

II **I**

Figure 6: Three-dimensional models created by Pro-E and created mesh by ABAQUS

Net production process is applied on the created model with the use of Fire software. Figure 7 demonstrates it in different angles.



II **I**

Figure 7: view of generated three-dimensional network in the 360 and 540-degree crankshaft angle

As it is shown in the figure, the simulation is done in an open cycle in which entrance valve timing is considered. The total number of the network at this stage is about 377000 cells. After the closure of the entrance valve, compression-expansion course is carried out. Finally, the simulation continues until the moment of outlet valve opening, which is equivalent to 840 degrees of crankshaft. To achieve sufficient accuracy for reaching to logical answer based on network focus, total network cells reaches to 50000 cells at the moment of ignition, which valves are near the high point of pause. Fame Engine Plus order is used for this procedure. FEP is a tool for networking and automated process to create mobile network.

[DOI: 10.22068/ase.2020.482]

Table 1: The initial conditions for modeling

Crankshaft angle at the start of calculation	360
Crankshaft angle at the end of calculations	840
Initial temperature-pressure	constant
Engine speed	1500 RPM
Type of hydrocarbon fuel	$C_{1.2}H_{4.4}$

The purpose of this research is to improve national gas engine87D. The gas engine87D is an offshoot of national engine D, which is under simulation by Diesel Sangin Iran Company. This engine is evaluated based on the effect of different coefficients rotation of fluid flow inside the combustion chamber through examining the shape of entrance port. Characteristic of this engine is provided in table 2:

Table 2: Characteristics of base engine

Functional Specification	Amount
Diameter cylindrical	150 mm
Course duration	180 mm
Chamber volume	38 liter
Compression ratio	15
Revolution	1500 RPM
Power	1000 KW
Ignition timing (gas engine)	15 BTDC

2.3. Sensitivity analysis

To search for networks sensitive to the network density, the results of two computational networks, with the number of different networks, is compared. In the large network the number of computational cells in high pause point is 377000, while, in the small network, the number of computational cells is 653000. The volume of these cells could not be smaller because the current systems are not able to load them at that high volume. As is clear in the diagrams, fragmenting cells will not have significant impact on the results. Also, computational network does not show much sensitivity to the size of cells. Therefore, large network was used to reduce the execution time.

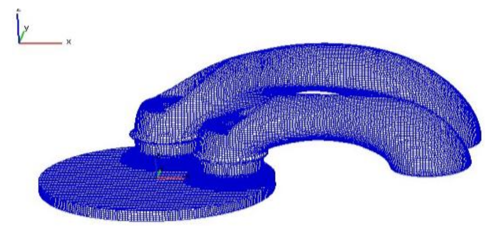


Figure 8: Networking with 377,000 cells per high-pause point

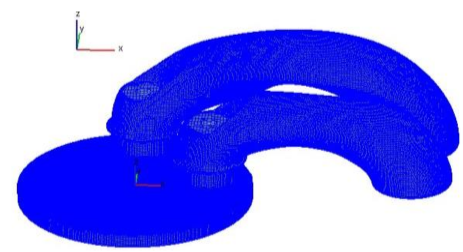


Figure 9: Networking with 653000 cells per high-pause point

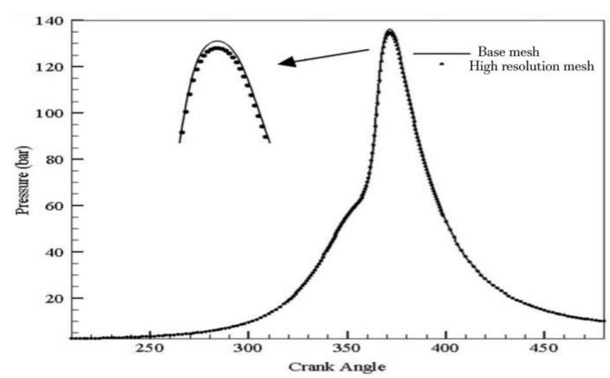


Figure 10: Pressure based on shaft degree for two different volumes in the networking

Gas engine 87 d has not yet produced and its experimental data is not available so far. In this regard, Cummins engine which has the same production capacity per the same volume was used to validate the generated model. The comparison for pressure changes characteristic inside the cylinder is presented in Figure 11. In fact, these two engines have the same class production power. After validation of generated model with the use of AVL FIRE software, and homogenization of coefficients, simulation for targeted engine will be initiated.

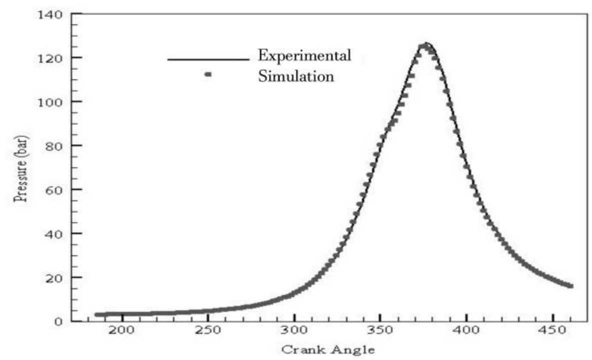


Figure 11: comparison between pressure chart resulted from geometry simulation of basic engine with experimental results

4. Numerical results

This research seeks to discover the effect of rotation coefficient on a gas engine that uses natural gas. For this purpose, coefficients of simulations as well as coefficients from different sources were selected to meticulously show that the range of this simulation is able to demonstrate the effect of rotation speed on engine performance. The examined coefficients are as follow: 0/26, 0/47 (basic engine), 0/68, 1, 1/3, 2, and 3.

Figure 12 indicates pressure changes inside the chamber based on the changes in coefficient rotation within the chamber. There is not significant change in peak pressure of inside the chamber when the rotation coefficient increases. But, when the speed reaches up to 3000 RPM (equivalent to 2 rotation coefficient), increase in peak pressure is clearly observable.

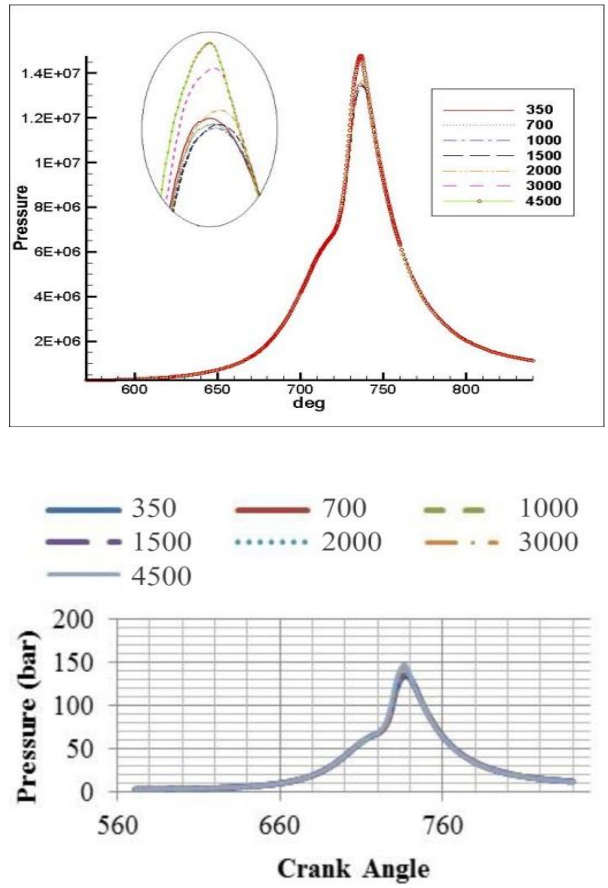
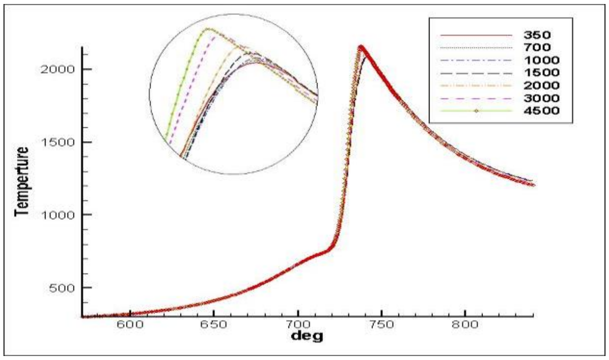


Figure 12: Changes in pressure inside the chamber

Figure 13 indicates temperature changes inside the chamber during compression and expansion course which is similar to pressure conditions. As expected, changes only occur for conditions with more than 2 rotation coefficient. As shown in figure 14, there are not any changes in different cycles. Also, the total amounts of injected energy through gas are provided at the same level, in all scenarios. It is possible to predict the reasons of pressure changes through a comparison between figure 14 and 15. As it is shown in the two figures, by increasing the fluid rotational speed inside the chamber, the speed of flame is also increase which leads to increase in the rate of heat release. Faster speed in release rate causes an increase in peak pressure. However, it is possible that output power does not show improvement due to the change happening at the peak moment.

[DOI: 10.22068/ase.2020.482]



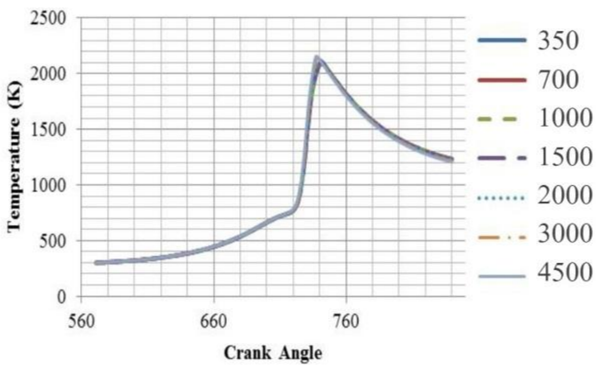


Figure 13: temperature changes inside the chamber

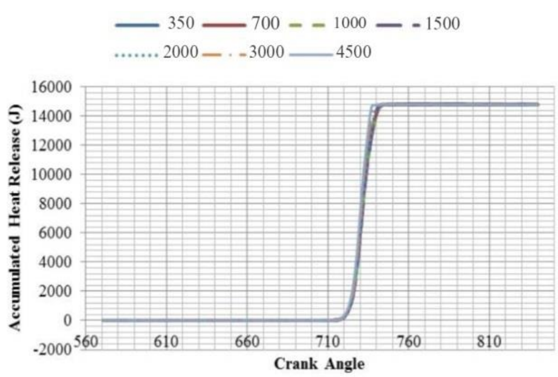


Figure 14: the total amount of energy injected into the chamber

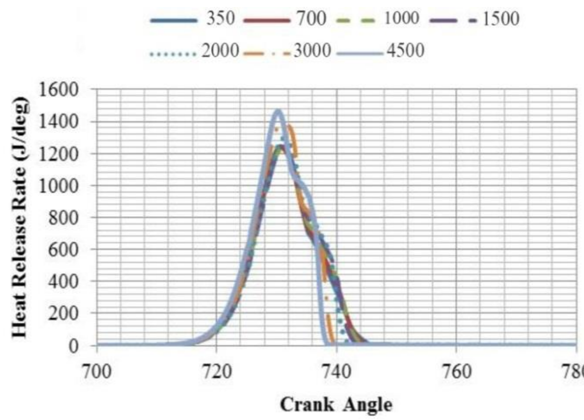


Figure 15: Heat release rate in degrees crankshaft

Figure 16 demonstrates the velocity distribution (in different parts) inside the combustion chamber at the moment of entrance valve closure. By increasing the rotation speed, the lines density increases (while the current around the vertical axis are becoming more willing to flat line). More precisely, as it is apparent in the first image on the left, 370 RPM rotation was around perpendicular to the center of current. The last contour on the right shows the current speed for 4500 RPM. Figure 17 shows the same result for the moment of maximum pressure (about a 740 angle which is 20 degree posterior to high density point).

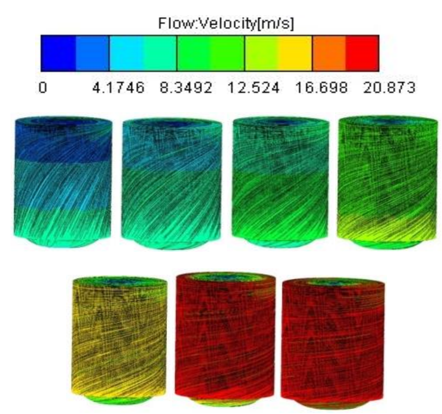


Figure 16: Velocity distribution and flow lines for different rotation coefficient on the angle of entrance valve closure

[DOI: 10.22068/ase.2020.482]

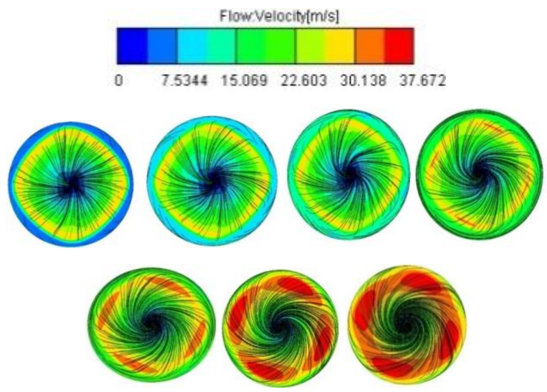
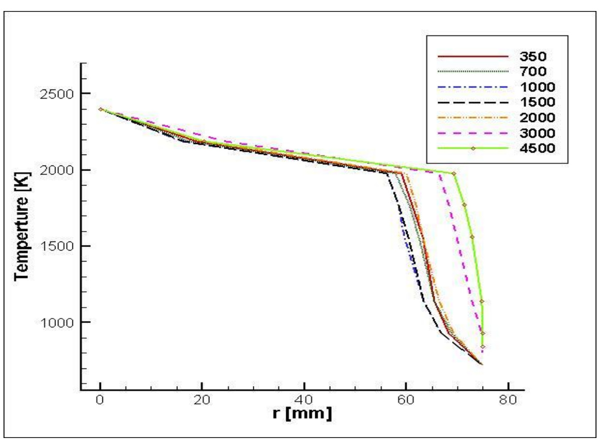
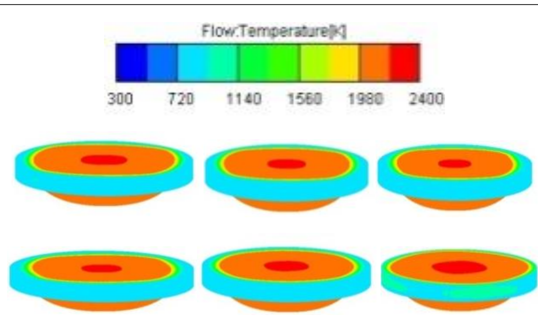


Figure 17: Velocity distribution and flow lines for different rotation coefficient on the angle of entrance valve closure

Figure 18 shows the temperature distribution for the crankshaft at an angle of 740. As it is observable in these contours, by increasing rotation coefficient and concomitantly increasing the combustion speed, temperature at higher rotation speeds reaches the walls sooner and the flame is distributed over a shorter period of time.





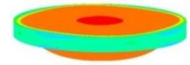
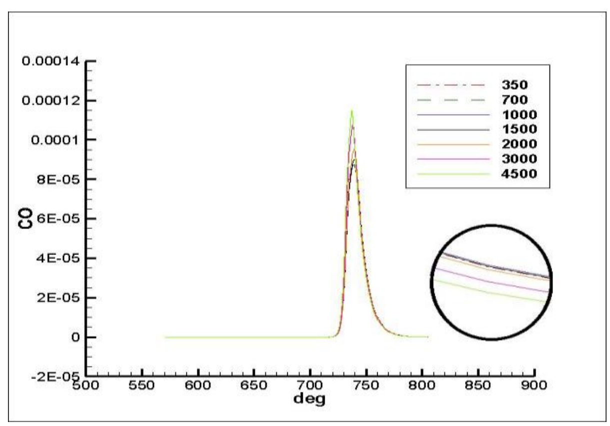


Figure 18: Distributed Temperature 20 ° after ignition

According to changes in flame speed indicators, the amounts of pollutants have been affected. Carbon monoxide emissions are generated during conversion of CO to CO2 due to lack of enough oxygen at the time of fuel combustion. Carbon monoxide emissions decrease with a fixed rate when flow rate (rotation coefficient) increases (figure 19). This event, due to increase in current flow inside the chamber and availability of oxygen to different layers, is combustion which has more collision surface with oxygen at different moments, therefore, results in better combustion. Figure 20 demonstrates the mount of unburned fuel at the opening moment of outlet valve which causes unburned fuel to be released to outside environment. As it is obvious in the figure, the amount of unburned fuel significantly reduced for rotation coefficient more than base case. So that, its rank is higher than 10th level based on the rotation speed around 400. For this reason, other speeds, except low speeds, are demonstrated within the range of almost zero.



[DOI: 10.22068/ase.2020.482]

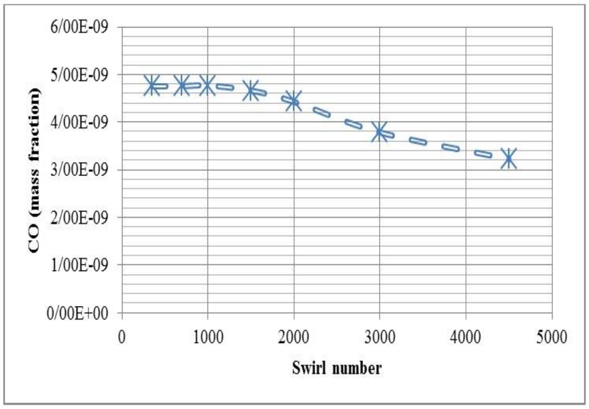


Figure 19: rate of carbon monoxide emissions

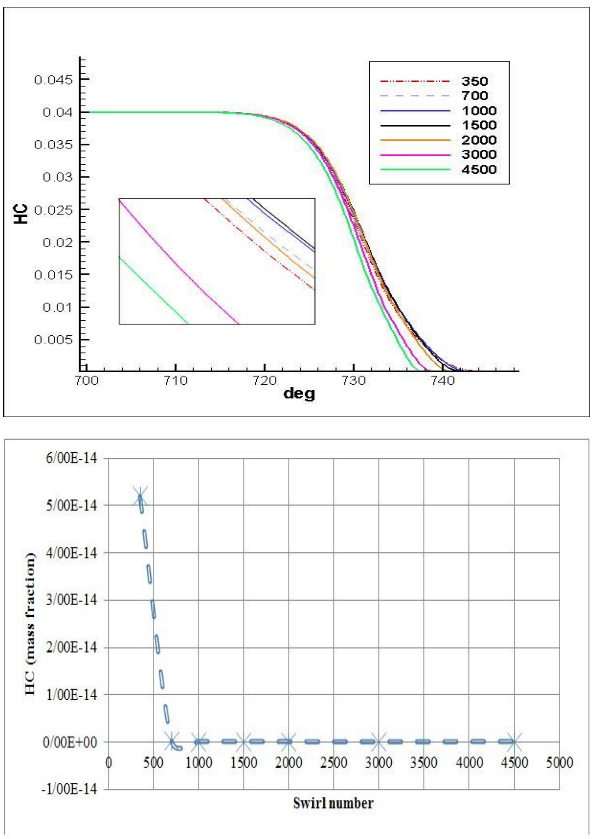


Figure 20: the amount of unburned fuel

Temperature increase inside the chamber and the amount of available oxygen are two factors that directly have an effect on the rate of carbon monoxide emissions. As we saw in the temperature curve, speed increase leads to increase in temperature. In this regard, figure 21 shows that there is not any certain change in the pollutants production in speed rotation less than 1500. However, by increasing the speed, the amount of pollutants increases. This issue across all surfaces inside the chamber is perfectly observable in figure 22. In middle parts, where flame progress occurs because of the candle, has higher temperature. Therefore, there are more pollutants in these areas.

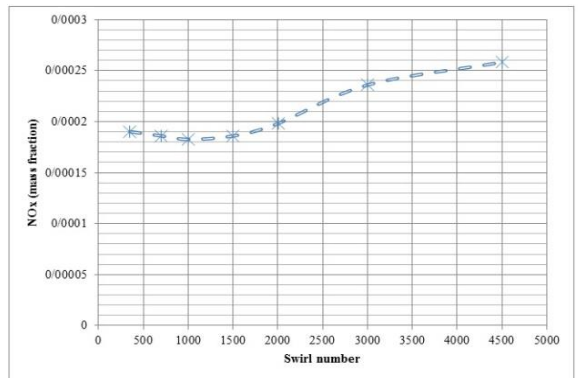


Figure 21: Nitrogen oxide emissions rate

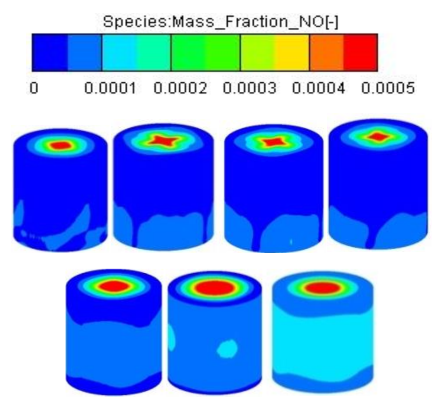


Figure 22: Distribution of nitrogen oxide emissions

There is a negative correlation between change rate of two pollutants carbon monoxide and nitrogen oxide. Thus, an optimal point selection is possible only if two graphs are considered at once. Approximately, rotation coefficient was 2200 meeting points between the two graphs. They can be used as the optimal functional engine.

[DOI: 10.22068/ase.2020.482]

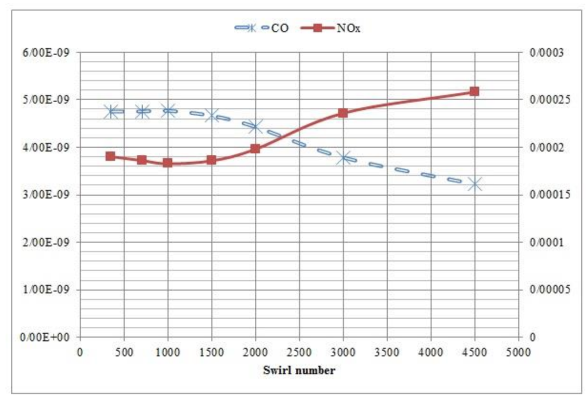


Figure 23: Comparison between pollutants of carbon monoxide and nitrogen oxide

The change of power based on changes in other engine's factor is presented in figure 24. Bar charts show that there is not significant change in production capacity when rotation coefficient increases. According to the presented amounts, the highest production capacity with 3000 RPM rotation is less than 2 kilowatts higher than its minimum value.

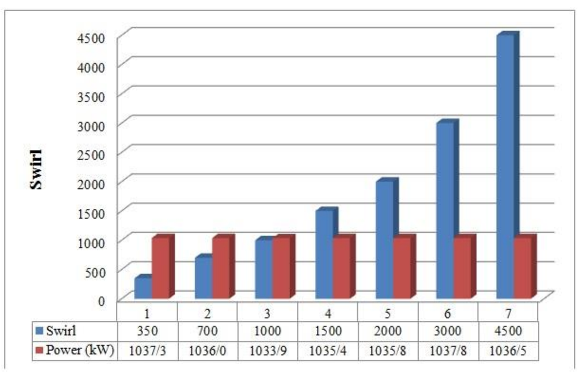


Figure 24: Production capacity

5. Conclusion

General conclusion of current study is presented as follow:

- Increasing rotation coefficient less than 1 causes insignificant change in maximum pressure and temperature inside the chamber while more rotation coefficient increase leads to observable significant change especially on 2 and 3 rotation coefficient.
- Despite the change in pressure graph, which is due to shifting peak pressure timing and stability of volume pressure under the diagram, power changes inside the chamber is minimal with little swing in comparison with the base case.
- Carbon monoxide emissions and unburned hydrocarbons have changed under the influence of considered indicator. This indicator has caused the two pollutants to decrease significantly when rotation coefficient increases.
- As one of the most toxic products of combustion, nitrogen oxide emissions decreases based on the temperature decrease inside the chamber. On the other hand, nitrogen oxide increases based on changes in fluid mixture, especially fuel and air. Conditions such as carbon monoxide emissions or output power should be considered to select the optimal performance point. With the consideration of these conditions, the optimal performance point shows fluid rotation speed with about 2000 RPM.

References

[1] J. Kawashima, Research on a variable swirl intake port for high-speed 4-valve DI diesel engine, JSAE Review 20, (1999), 421438.

[2] Kern, y. Kang, Rolf D. Retz, The Effect of Intake Valve Alignment on Swirl Generation in a DI Diesel engine, Engine Research and Development, Korea IMM, ERC, University of Wisconsin-Madison, (1999).

[3] V. PRASAD, S.L. PANDURANGADU, V. PRATIBHA BHARATHI, V.V. NAGA DEEPTHI, Experimental Study Of The Effect Of In Cylinder Air Swirl On Diesel Engine Performance, International Journal of Engineering Science and Technology (IJEST), ISSN, 0975-5462, Vol. 3 No. 2 Feb, (2011).

[4] M.L.S. Deva Kumar, S. Kamalesh Rao, Effect of Cylinder Air Swirl on Petrol Engine Performance, International Journal of Applied Research & Studies ISSN 2278 – 9480, iJARS/ Vol.I / Issue II /Sept-Nov, (2012-219).

[DOI: 10.22068/ase.2020.482]

[5] V.v. Prathibha Bharathi, Dr. G.Prasanthi. The Influence of Air Swirl on Combustion and Emissions in a Diesel Engine, Dept, of Mechanical Engineering, MREC (Autonomous), Hyderabad, AP, India. IJRMET Vol. 3, Issue 2, (2013), May – Oct.

[6] B. Jafari, D.Domiri Ganji, Numerical Investigation in the Effect of Number of Nozzle Hole on Performance and Emission in Dual Fuel Engine, International Journal of Automotive Engineering Vol. 3, Number 2, June 2013.

[7] H. Biglarian, S. M. Keshavarz, M. Sh. Mazidi1, F. Najafi, A Strategic Mild Hybrid Technology for Reducing Pollution and Optimization of Fuel Sources, International Journal of Automotive Engineering Vol. 4, Number 4, Dec 2014.

[8] Moein Nili Ahmadabadi, Pouria Ahmadi, Mahdi Soleymani, Seyed Alireza Atyabi, Mohammadjafar Hadad, Environmental Impact Assessment of replacing Conventional Taxis with Hybrid Electric Vehicles in Tehran, Iran, International Journal of Automotive Research, Vol. 9, No. 4, (2019), 3125-3131

[9] AVL FIRE CFD Solver V. 8.3, 2008.

[10] AVL FIRE Combustion V. 8.3, 2008

[DOI: 10.22068/ase.2020.482]