

3D COLD FOLLOW SIMULATION INSIDE INTAKE MANIFOLD AND CYLINDER OF AN IC ENGINE

Bilge Albayrak ÇEPER*, Melih YILDIZ*, Vedat DEMIRTAS**,
Hakan Serhad SOYHAN***, Nafiz KAHRAMAN*

*Erciyes University, Faculty of Engineering, Department of Mechanical Engineering, 38039 Kayseri, Turkey

** Afyon Kocatepe University, Faculty of Technology, Dept. of Automotive Engineering, 03200 Afyonkarahisar

*** Sakarya University, Faculty of Engineering, Department of Mechanical Engineering, 54187 Sakarya

*Corresponding author: E-mail: balbayrak@erciyes.edu.tr Tel: +90 352 2076666, Fax: +90 352 4375784

Abstract

This paper presents 3D cold follow simulation inside the intake manifold, valves and cylinders of the engine which has two cylinders with two valves in each, and a compression ratio of 10.7. The purpose of this study is to analyze the change in the cylinders in terms of temperature, pressure and velocity depending on crank angle degree for 300 K and 450 intake temperatures. This is, particularly, important for a controlled auto ignition (CAI) engine which has sensitive to temperature distribution and pressure for combustion development in cylinders. In this study, the engine model based on the real dimension of Lombardini LGW523 engine was structured and then meshed for follow fields. Fluent CFD code which uses finite volume method was used as an analysis tool. 3D Pressure based Implicit unsteady solver was used to solve the basic governing equations (mass, momentum and energy). RNG k- turbulence model was used to simulate the turbulence follow in the cylinders. The analyses in this study were carried out for four strokes of the engine using dynamic mesh. As a result, in cylinder temperature, pressure and velocity distribution were obtained with change in crank angle positions and compared for two different intake temperature.

Keywords: Cold flow simulation, Spark Ignition Engine, Intake manifold, Cylinder Pressure

1. INTRODUCTION

An inlet manifold or intake manifold is the part of an engine that supplies the fuel/air mixture (in case of SI engine) or only fresh air (in case of CI engine) to the engine cylinder. The primary function of the intake manifold is to evenly distribute the combustion mixture (or just air in a direct injection engine) to each intake port in the cylinder heads[1]. Improvements in engine efficiency can also be attained by enhanced in-cylinder flow motions using optimized cylinder head, port and valve timing. A well-designed intake manifold and cylinder head will reduce the flow resistance and increases the swirl. Swirl is the rotation of charge about cylinder axis and it is used in SI engines to speed up the combustion[2]. In-cylinder flow field structure in an internal combustion (IC) engine has a major influence on the combustion, emission and performance characteristics. The fluid motion in an internal combustion engine is induced during the induction process and later modified during the compression process. Intake charge enters the combustion chamber through the intake manifold of an IC engine with high velocity. Then the kinetic energy of the fluid resulting in turbulence causes rapid mixing of fuel and air, if the fuel is injected directly into the cylinder. In-cylinder fluid motion governs the flame propagation in spark-ignition engines, and controls the fuel-air mixing and premixed burning in diesel engines. With optimal turbulence, better mixing of fuel and air is possible which leads to effective combustion [3].

Paul and Ganesan [4] studied the effect of helical, spiral, and helical-spiral combination manifold configuration on air motion and turbulence inside the cylinder of a Direct Injection (DI) diesel engine motored at 3000 rpm. Three dimensional model of the manifolds and the cylinder was created and meshed using the pre-processor GAMBIT. The flow characteristics of these engine manifolds were examined under transient conditions using Computational Fluid Dynamics (CFD) code STAR-CD. The predicted CFD results of mean swirl velocity of the engine at different locations inside the combustion chamber at the end of compression stroke were compared with experimental results carried out by other researchers. They concluded that swirl ratio inside the cylinder and turbulent kinetic energy was higher for spiral manifold. Volumetric efficiency for the spiral-helical combined manifold was 10% higher than that of spiral manifold.

Kumar and Nagarajan [5], deals with experimental investigation of swirling flow in the cylinder of a spark ignited engine. The variations in different non-dimensional parameters such as flow coefficient, swirl coefficient and swirl ratio at various valve lifts and throttle opening was studied using a steady flow bench. Their results indicate that a higher swirl coefficient and swirl ratio could be achieved with shrouded valve and twisted tape with penalty on flow coefficient. These experimental results could be utilized to verify the results of modeling using commercial CFD codes and also for increasing the

accuracy of mathematical modeling of combustion process in internal combustion engines.

Karthikeyan et al.[6] indicated from the AVL BOOST software that the sudden increase in pressure waves are observed with initial manifold design. Therefore the initial intake manifold was optimized for uniform flow, by using CFD software. From the CFD results, 76% mass fraction of air was observed for all the three runners at 1800 rpm. Further experimentally air pressure inside the runners were investigated and increased air pressure of 13% shows that flow of air increased inside the runner for the optimized intake manifold design.

The manifold design and temperature distribution in a cylinder is also important for a CAI engine which is alternative combustion mode to conventional SI and CI engines. This is because CAI combustion takes place at many points in a cylinder. During CAI combustion, premixed fuel and air mixture is compressed to a temperature so high (usually 1000 K or more) that auto ignition happens[7]. In case of high ignition temperature, we have to increase the in-cylinder temperature by means of heating intake port, modified compression ratio and using internal exhaust gas recirculation. Therefore, intake temperature value, temperature distribution and mass fractions of mixture directly influence on CAI combustion process. In this study, 3D flow

simulation was performed at two different intake temperatures for CAI engine converted from SI engine in order to attain temperature distribution in the flow fields.

2. NUMERICAL ANALYSIS

The specifications of the SI engine analyzed are given in Table 1. A geometric model of the engine including intake manifold, combustion chamber, intake and exhaust ports, intake and exhaust valves, was structured and meshed for computational domains by using GAMBIT. The computational mesh consisted of 2.368.713 cells at TDC and 5.605.462 cells at BDC. Figure 1 shows the computational domain of the three-dimensional geometric model.

Table 1. Engine Specification

Description	Unit	Value
Bore	[mm]	72
Stroke	[mm]	62
Displacement volume	[lt]	0.505
Compression ratio	[-]	10.7
Intake valve diameter	[mm]	32
Exhaust valve diameter	[mm]	27

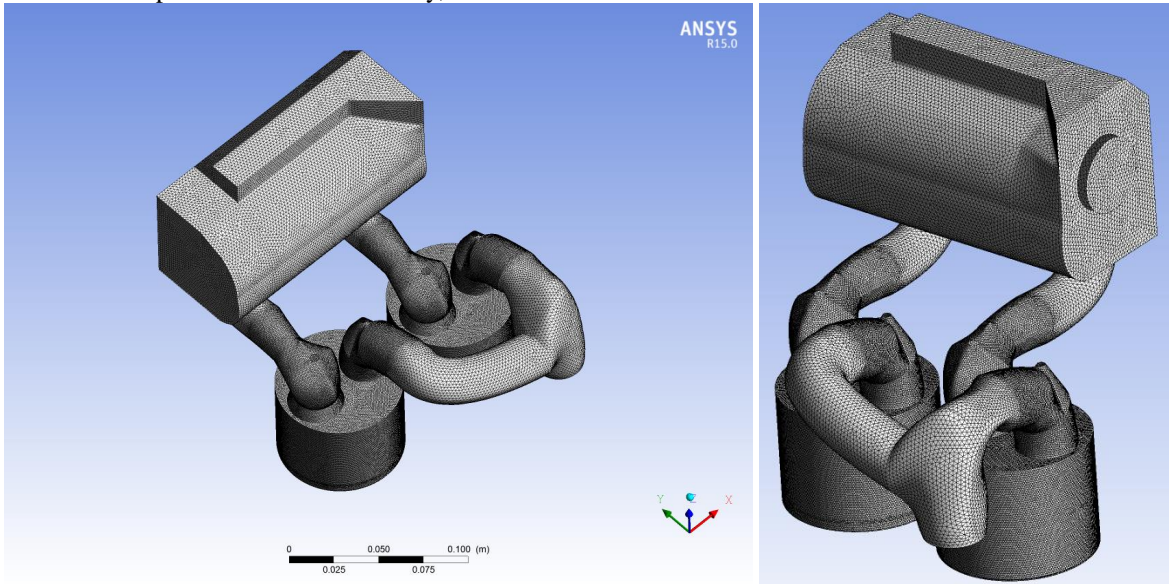


Figure 1. Geometric model.

Model formulation:

The model considered was solved using FLUENT which uses the control-volume-based method to convert the governing equation to algebraic equations. The governing equations of continuity, momentum, energy and the sub-model equations of turbulence, radiation and species transport with finite

rate chemistry, auto-ignition and NO_x model, which are necessary to simulate the engine cycle were solved in time dependent deforming two-dimensional control volume. The integral form of the conservation equation for ϕ scalar quantity is written for an arbitrary control volume V as follows [8]:

$$\int_V \frac{\partial \rho \phi}{\partial t} dV + \oint \rho \phi \vec{v} \cdot d\vec{A} = \oint \Gamma_\phi \nabla \phi \cdot d\vec{A} + \int_V S_\phi dV \quad (1)$$

where ρ is density, \vec{v} is velocity vector, \vec{A} is surface area vector, Γ_ϕ denotes diffusion coefficient for ϕ and S_ϕ represents source of ϕ per unit volume.

A. Turbulence Model

The RNG k-ε turbulence model was used owing to its degree of accuracy in obtaining the effect of swirling flow. Transport equations for the RNG k-ε are expressed as follows:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}(\rho k u_j) = \frac{\partial}{\partial x_j} \left(\alpha_k \mu_{eff} \frac{\partial k}{\partial x_j} \right) + G_k + G_b - \rho \epsilon - Y_M + S_k \quad (2a)$$

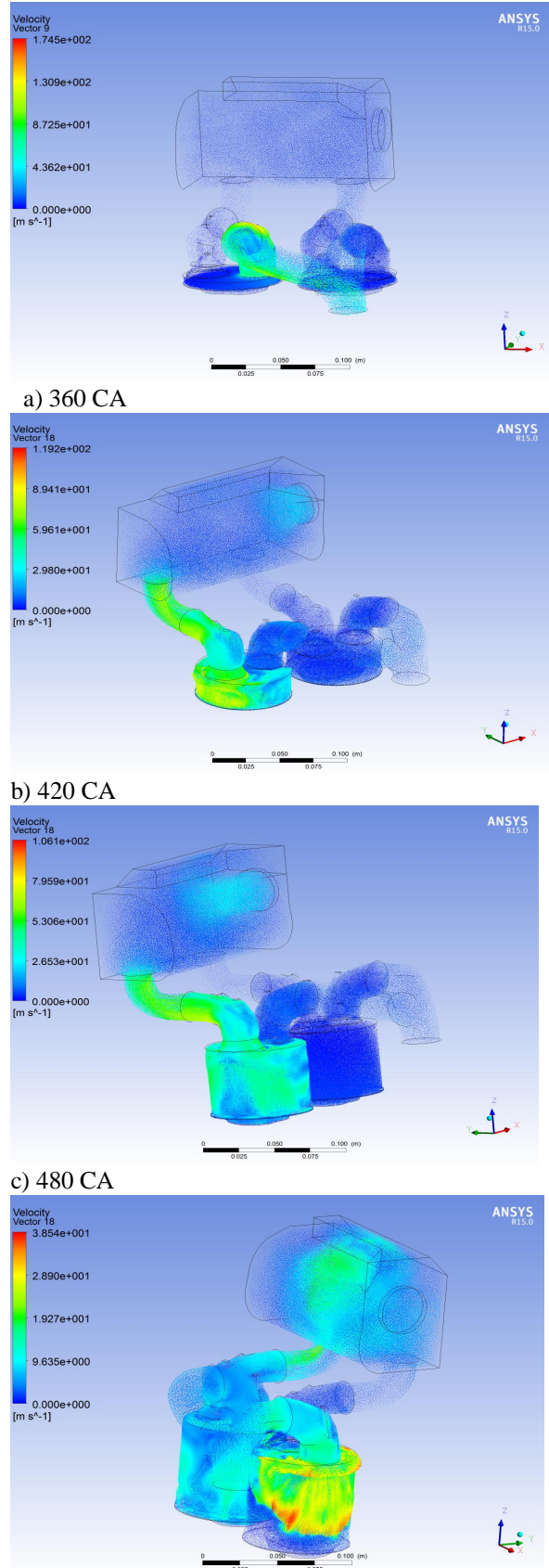
$$\frac{\partial}{\partial t}(\rho \epsilon) + \frac{\partial}{\partial x_j}(\rho \epsilon u_j) = \frac{\partial}{\partial x_j} \left(\alpha_\epsilon \mu_{eff} \frac{\partial \epsilon}{\partial x_j} \right) + C_{1\epsilon} \frac{\epsilon}{k} (G_k + C_{2\epsilon} G_b) - C_{2\epsilon} \rho \frac{\epsilon^2}{k} - R_\epsilon + S_\epsilon \quad (2b)$$

In these equations, G_k and G_b denote the generation of turbulence kinetic energy due to mean velocity gradients and buoyancy, respectively. Y_M is the contribution of fluctuating dilatation in compressible turbulence. α_k and α_ϵ are the inverse effective Prandtl number for k and ϵ . S_k and S_ϵ are user-defined source terms. $C_{1\epsilon}$ and $C_{2\epsilon}$ are the model constants derived analytically by the RNG theory.

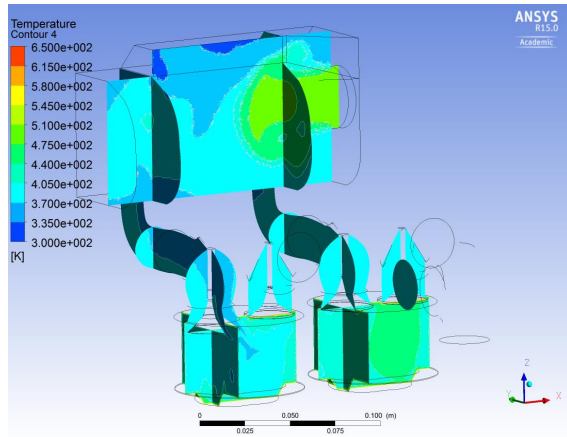
The total temperature is set to 300 K for case one, and then 450 K for case two, assuming that the engine is naturally aspirated. Three main boundary features were used (i) pressure inlet (ii) pressure outlet and (iii) walls. Turbulence in the related boundaries was specified using Intensity and Hydraulic Diameter. Wall temperatures were also set to 300 K. 3D Semi Implicit unsteady solver was used to solve the basic governing equations (mass, momentum and energy). RNG k-ε turbulence model was used to simulate the turbulence in the engine cylinder. SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) was used as the pressure velocity coupling scheme.

3. RESULTS AND DISCUSSION

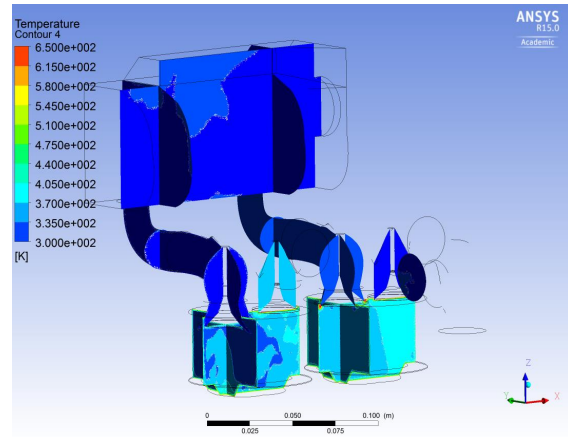
The results from the manifold and cylinders modeling and CFD simulation using FLUENT software are shown and discussed. Figure 1 shows the contours of velocity magnitude at different crank angles. As seen in this figure, velocity magnitudes are clearly seen in manifold and cylinder with valve. Piston is in TDC position where the exhaust valve is closed and the inlet valve is open i.e., beginning of intake stroke for first cylinder. Maximum velocity obtained at 540° is 38.54m/s.



d) 540 CA
Figure 1. Velocity magnitude for different crank angles

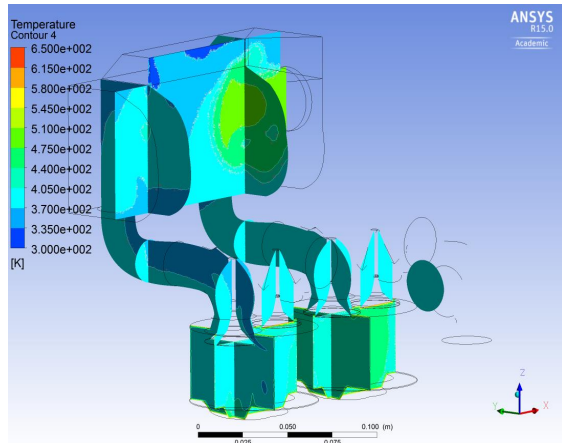


a) 450 K at 450CA

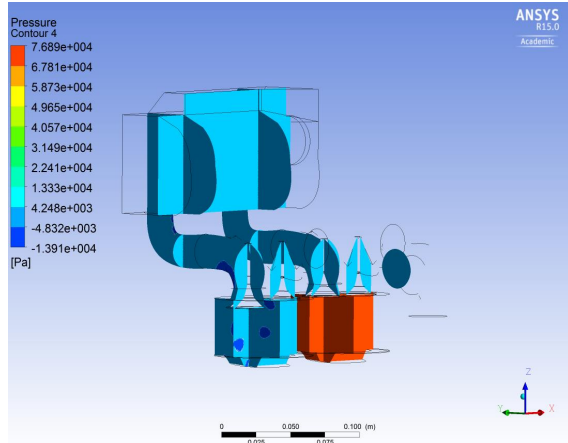


b) 300K at 450CA

Figure 2. Temperature distribution for 300 and 450 K at 450 CA



a) 450 K at 450CA



b)300K at 450CA

Figure 3. Pressure distribution for 300 and 450 K at 450CA

Figure 4 shows pressure values versus the crank angle for 300 and 450 K intake temperature. With the

increasing of intake temperature, pressure values were decrease in both cylinders.

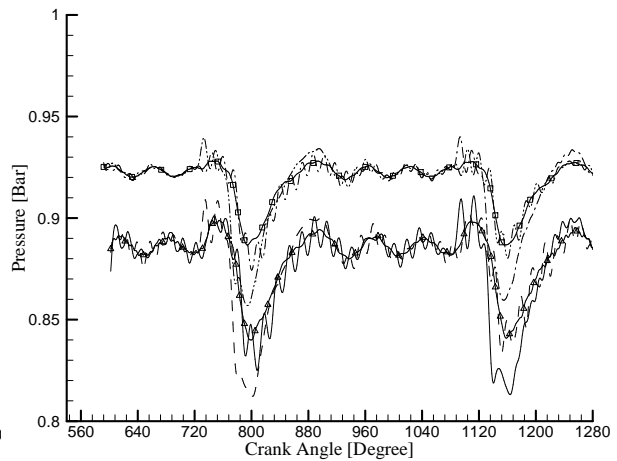
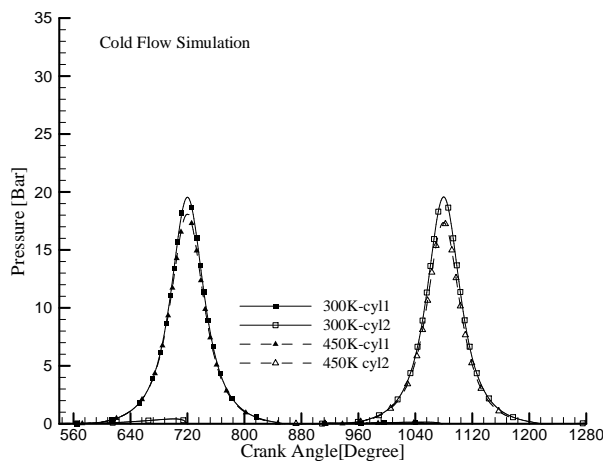


Figure 4. Pressure values versus the crank angle for 300 and 450 K intake temperature.

The initial intake manifold is not able to provide uniform air distribution to all the cylinders for CAI combustion mode(heating intake temperature). This

leads to decrease the performance of the engine. Therefore, this manifold is changed different configuration relative with CAI engine mode.

4. CONCLUSIONS

The effect of intake temperature on controlled auto-ignition (CAI) engine considering the two different intake temperature by using the CFD code FLUENT program. RNG k- model was used to simulate the turbulence in the engine cylinder. SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) was used as the pressure velocity coupling scheme. All four strokes and their effect on in-cylinder air motion were studied effectively using the numerical approach followed in this paper. CFD can be used as a useful tool to determine the effect of the parameters related to engine on performance.

ACKNOWLEDGMENTS

The authors of this paper would like to acknowledge the financial support provided by TUBITAK Turkey under contract number 113M101.

References

- [1] Shah J.V., Patel P.D., Jotava D.J., Effect of Intake Manifold Inclination on Performance and Emission Parameters of 4-Stroke Single Cylinder C.I. Engine: A Technical Review, Int J of Engineering Research & Technology (IJERT), Vol. 3 Issue 1, January - 2014.
- [2] Heywood J.B., Internal Combustion Engine Fundamentals. McGraw-Hill, New York, USA, 1988.

- [3] Murali Krishna B., Bijucherian A., and Mallikarjuna J. M., Effect of Intake Manifold Inclination on Intake Valve Flow Characteristics of a Single Cylinder Engine using Particle Image Velocimetry, World Academy of Science, Engineering and Technology, 44, 2010.

- [4] Paul B., Ganesan V., Flow field development in a direct injection diesel engine with different manifolds, International Journal of Engineering, Science and Technology Vol. 2, No. 1, pp. 80-91, 2010.

- [5] Kumar C.R. and Nagarajan G., Investigation of Flow During Intake Stroke of a Single Cylinder Internal Combustion Engine, ARPN Journal of Engineering and Applied Sciences, vol 7 no. 2, 2012.

- [6] Karth, keyan, S., Har, ganesh, R., Sathyanadan, M., Krishnan, S., Vadivel, P., Vamsidhar, D., Computational analysis of intake manifold design and experimental invest, gat, on on diesel engine for LCV. International Journal of Engineering Science and Technology (IJEST). Vol. 3 No.4, pp- 2359-2367, 2011.

- [7] Li J., Zhao H., Ladommatos N., Ma T., Research and Development of Controlled Auto-Ignition (CAI) Combustion in a 4-Stroke Multi-Cylinder Gasoline Engine, SAE Technical Paper 2001-01-3608, 2001.

- [8] Fluent 6.3 User's Guide, 2006. Fluent Incorporated, Centerra Resource Park, 10. Cavendish Court, Lebanon, NH 03766, USA.