IOP Conference Series: Materials Science and Engineering

PAPER • OPEN ACCESS

Investigation of Discharge Flow and Force Coefficients in Hermetic Reciprocating Compressors

To cite this article: Caglar Sahin *et al* 2019 *IOP Conf. Ser.: Mater. Sci. Eng.* **604** 012026

IOP Conf. Series: Materials Science and Engineering **604** (2019) 012026 doi:10.1088/1757-899X/604/1/012026

Investigation of Discharge Flow and Force Coefficients in Hermetic Reciprocating Compressors

Caglar Sahin[1], Hüsnü Kerpicci[1], Ahmet Yasin Karabay[1], Kerem Karahan[1]
1 Arçelik A.Ş. R&D Directorate 34950 Tuzla İstanbul Turkey

Corresponding author: +902165858319
E-mail: caglar_sahin@arcelik.com

Abstract. Low energy consuming products are eternal target of all household appliance manufacturers. To acquire desired energy index, high efficiency variable capacity compressors have started to be used widely on refrigerators. Reed valves and valve plate designs are the most critical issues in the development of reciprocating compressors in terms of coefficient of performance (COP). The stiffness of the reed valves and the port geometries play an important role on reducing the losses at various speeds and operating conditions. In the last decade, Fluid-Structure Interaction (FSI) method has started to be used in order to investigate valve dynamics. Because of the complexity and high computational cost of FSI method, system simulation tools are still preferred for faster and simpler solutions.
In-house developed simulation tool uses flow and force coefficients to calculate valve dynamics and mass flow rate. Those are calculated with the Computational Fluid Dynamics (CFD) analyses with respect to the valve lift. In the present study, flow and force coefficients at different piston positions were investigated. Piston pin which is used to reduce dead volume was also considered as a parameter. CFD calculations were established for steady state conditions at different valve positions with Ansys Fluent. The calculated flow and force coefficients were implemented into the simulation tool and the effect of piston position was presented in terms of COP, cooling capacity and compared with the experimental results.

1. INTRODUCTION

Geometries of discharge and suction valves are important design parameters of the compressor in terms of efficiency. Generally, parametric analyzes are required to determine optimum port and valve geometries for new compressor design. FSI approach can be used for as an as ideal method to understand the valve dynamics. However, because of its complexity and high computational cost it is not applicable for parametric analysis. Instead of FSI, lumped simulation models are widely used in literature to determine design parameters.

Kerpicci and Oguz (2006) investigated the mass flow responses of the series orifice model employing the flow coefficients derived from the steady state and transient computational fluid dynamics (CFD) analysis for different valve lifts. A dynamic pressure difference is applied through the port and the valve leaf for both cases. The error associated with the time integrated average mass flow rate calculation was found to be on the order of 5% for the specific geometry and pressure pulsation profile.

Pereira and Deschamps (2010) investigated the influence of the piston on the effective flow and force areas of a simplified geometry of discharge valve. A numerical model based on a two-dimensional formulation was developed to simulate the discharge process for different combinations of valve lift and piston positions. Through results for effective flow and force areas, the analysis made it evident that the piston position in relation to the cylinder head should be included to fully characterize the valve performance. However, 2-D CFD analyzes were performed and no specific arguments on the model results was cited in this paper.

Sergio et. Al. (2018) investigated the effect of mass flow inertia and backflow on the effective flow and forces areas in a lumped simulation model. The variation related with using different effective flow and force areas for normal and backflow conditions on the valve impact velocity was up to almost 19%. Also taking into consideration mass flow inertia terms decreased the backflow up to 100%.

Guangyu et. al. (2019) proposed a new one degree of freedom model to analyze valve dynamics which includes the effects of viscous stiction and relationship between the effective length, elastic force and mass of the reed with the change of displacement. The model results were validated with the experimental measurements for the discharge side. The effect of valve stiffness, limiter height, orifice diameter and compressor speeds were investigated parametrically on pressure and velocity profiles.

In the present paper; the effect of piston position on the flow and force coefficients were investigated in CFD analyzes. In addition, the coefficients which were calculated based on the piston position and valve lift were implemented into the new 1-D model and calculated cooling capacity, COP and discharge losses results were compared with previous 1-D model and experimental results at ASHRAE conditions.

2. CFD ANALYSIS

The 3D geometry and cross-sectional view of the discharge flow line including cylinder and cylinder head internal volume were shown in Figure 1. The inlet and outlet boundary conditions were defined as pressure inlet (8.0bar, 400K) and the pressure outlet (7.6bar, 383K) respectively. The pressure inlet was set to the lateral surface of the cylinder since the radial flow is thought to reflect the actual situation better.

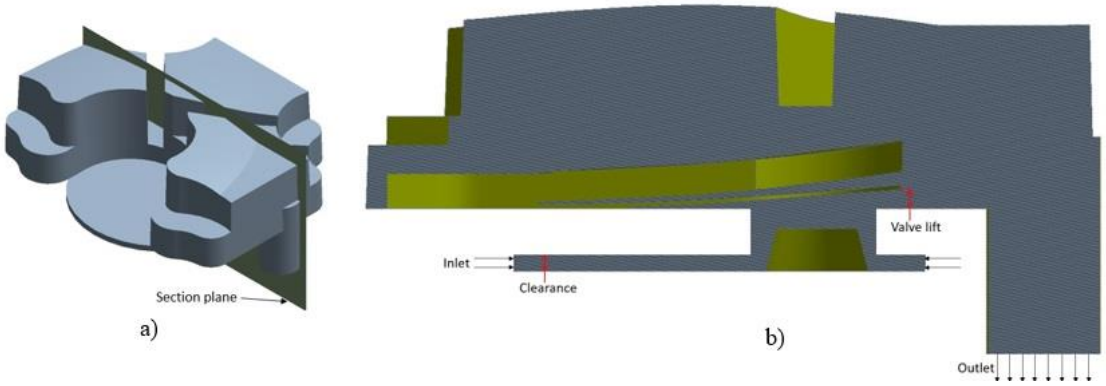

Figure 1. Isometric view (a) and section view (b) of 3D model.

Unstructured tetrahedral mesh was used in the computation domain. For each considered valve lifts and piston positions different meshes were generated. All meshes comprised about 10-12 million cells and the skewness values were kept <0.9. A sample mesh structure on the section plane was given in Figure2.

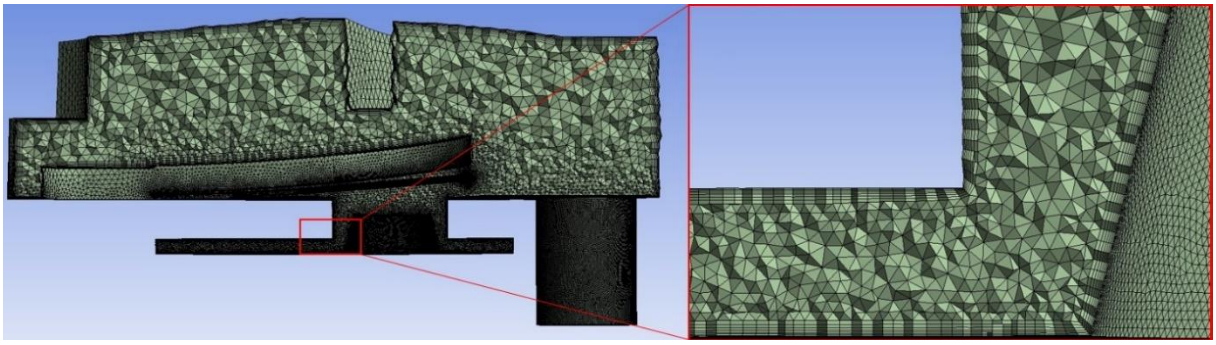

Figure 2. Mesh configuration of the section plane.

The analyzes were carried out with commercial software (Ansys, Fluent 19.2). Turbulent flow was solved by the Reynolds averaged flow equations. First-order upwind scheme was adopted in the solution of the momentum, energy and turbulence equations. The coupling between the pressure and velocity fields was achieved with the SIMPLE scheme. The realizable k-Ɛ approach was used to model turbulence and the standard wall function is applied with y+ > 11. Turbulence intensity was set to 7%. Isobutane with the ideal gas condition was used as fluid.

3. SIMULATION MODEL

1-D model was used to simulate piston/cylinder volume and valve dynamics. The flow in the cylinder was not solved and cylinder pressure was assumed to be homogenous. Cylinder wall temperature, discharge and suction plenum temperatures were taken constant and the data acquired from experimental measurements were used. The valves were modelled by dividing in several elements (nodal points) along the length of the valves. The mass of the valve was lumped in these discrete nodal points which were equally distributed over the valve length.

The refrigerant flow through the valve ports was described by the formula for incompressible flow, adapted for the compressible flow with an expansion coefficient. This yields the following set of equations:

$$\Phi = \alpha\varepsilon\omega ED\sqrt{2\rho(P_u - P_d)} \quad (1)$$

$$\varepsilon = 1 - \frac{c_v}{c_p} \cdot \left[\frac{P_u - P_d}{P_u}\right] \quad (2)$$

$$A_o = \omega ED \quad (3)$$

$$\alpha = \frac{\Phi_{cfd}}{\varepsilon A_o\sqrt{2\rho(P_u - P_d)}} \quad (4)$$

where; Φ is the mass flow rate, ε is the expansion coefficient, c_v and c_p are the specific heat at constant volume and constant pressure, respectively, ω is the valve lift, E is introduced as a circumferential scale factor (its nominal value = π), D is the port diameter, P_u and P_d are the upstream and downstream pressures, respectively, ρ is the refrigerant density and α is the flow coefficient which is a dimensionless number, A_o flow area, Φ_{cfd} is the mass flow which is calculated with CFD analysis. Flow coefficient represents the ratio of mass flow rate which is calculated from CFD analysis to the mass flow rate which is calculated from the compressible flow with an expansion coefficient. It is used to as a correction factor to eliminate the errors which come from 1-D control volume approximation in the equation (1).

The gas force acting onto the valve is described by Eq.5;

$$F_g = \boldsymbol{\alpha_f}\frac{\pi}{4}D^2(P_u - P_d) \quad (5)$$

$$A_v = \frac{\pi}{4}D^2 \quad (6)$$

$$\boldsymbol{\alpha_f} = \frac{F_{cfd}}{A_v(P_u - P_d)} \quad (7)$$

where; E_x is scale factor, D is the port diameter, P_u and P_d are the upstream and downstream pressures, α_f is force coefficient, A_v is the port area, F_{cfd} is the gas force which is calculated from CFD analysis. Force coefficient represents the ratio of gas force which is calculated from CFD analysis and gas force which is calculated from the pressure differences between upstream and the downstream. Similarly, it is used to as a correction factor to eliminate the errors which come from 1-D control volume approximation in the equation (5).

In the present 1-D model, flow coefficients are calculated only with respect to valve lifts. The flow and force coefficients are described by Eq.4 and 5 which are used in the present 1-D model;

$$\boldsymbol{\alpha} = \boldsymbol{A}\boldsymbol{w} + \boldsymbol{B} \tag{8}$$

$$\boldsymbol{\alpha_f} = \boldsymbol{E}\boldsymbol{w} + \boldsymbol{F} \tag{9}$$

where; $\boldsymbol{\alpha}$ and $\boldsymbol{\alpha_f}$ are flow and force coefficients respectively, A, B, E, F are the constant coefficients which are calculated with the CFD analyses, $\boldsymbol{w}$ is the valve lift.

In the new 1-D model, in addition to valve lifts crank angle which describes the piston positions, is also taken into account to calculate flow coefficients . The flow and force coefficients are described by Eq.6 and 7 which are used in the new 1-D model;

$$\boldsymbol{\alpha} = \begin{cases} S_{1,1}w^2 + S_{2,1}w + S_{3,1};\ \boldsymbol{\gamma_1} \le \widetilde{\boldsymbol{\phi}} \le \boldsymbol{\gamma_2} \\ S_{1,2}w^2 + S_{2,2}w + S_{3,2};\ \boldsymbol{\gamma_2} < \widetilde{\boldsymbol{\phi}} \le \boldsymbol{\gamma_3} \\ S_{1,3}w^2 + S_{2,3}w + S_{3,3};\ \boldsymbol{\gamma_3} < \widetilde{\boldsymbol{\phi}} \le \boldsymbol{\gamma_4} \\ S_{1,4}w^2 + S_{2,4}w + S_{3,4};\ \boldsymbol{\gamma_4} < \widetilde{\boldsymbol{\phi}} \le \boldsymbol{\gamma_5} \\ S_{BF,fl};\ \boldsymbol{\gamma_5} < \widetilde{\boldsymbol{\phi}} \end{cases} \tag{10}$$

$$\boldsymbol{\alpha_f} = \begin{cases} \mathbf{S_{1,5}w^2 + S_{2,5}w + S_{3,5}};\ \boldsymbol{\gamma_1} \le \widetilde{\boldsymbol{\phi}} \le \boldsymbol{\gamma_2} \\ \mathbf{S_{1,6}w^2 + S_{2,6}w + S_{3,6}};\ \boldsymbol{\gamma_2} < \widetilde{\boldsymbol{\phi}} \le \boldsymbol{\gamma_3} \\ \mathbf{S_{1,7}w^2 + S_{2,7}w + S_{3,7}};\ \boldsymbol{\gamma_3} < \widetilde{\boldsymbol{\phi}} \le \boldsymbol{\gamma_4} \\ \mathbf{S_{1,8}w^2 + S_{2,8}w + S_{3,8}};\ \boldsymbol{\gamma_4} < \widetilde{\boldsymbol{\phi}} \le \boldsymbol{\gamma_5} \\ \mathbf{S_{BF,fo}};\ \boldsymbol{\gamma_5} < \widetilde{\boldsymbol{\phi}} \end{cases} \tag{11}$$

where; $\boldsymbol{\alpha}$ and $\boldsymbol{\alpha_f}$ are flow and force coefficients respectively, $\mathbf{S_n}$ are the constant coefficients which are calculated with the CFD analyses, $\widetilde{\boldsymbol{\phi}}$ is the instant crank angle, $\boldsymbol{\gamma_n}$ are the crank angle which determines the valid range for each polynomial expression.

The flow and force coefficients were specified with steady state CFD analysis. The coefficients were calculated for different valve lifts and implemented to the present 1-D model. The piston was held at the position corresponding 340° crank angle for the CFD analysis. In the new 1-D model the coefficients were calculated for different valve lifts and piston positions therefore, piston position is concerned another parameter to specify related coefficients.

4. EXPERIMENTAL STUDY

In the experimental study, pressure transducer and optical encoder was used to obtain indicator diagram. The pressure transducer was placed in the valve plate to measure pressure in the cylinder. The optical encoder was assembled at the top of the compressor crankcase with an elastic coupling and special designed apparatus to measure crank angle simultaneously in accordance with the pressure transducer. Figure 3 shows the implementation of the optical encoder and the pressure transducer.

The transducer has an accuracy of 0.1% FSO due to non-linearity and hysteresis. Pressure range is 0–17 bar absolute with a wide temperature range of -55°C to +175°C. The optical encoder has an accuracy of ±0.8 arc-min and maximum frequency response of 200kHz.

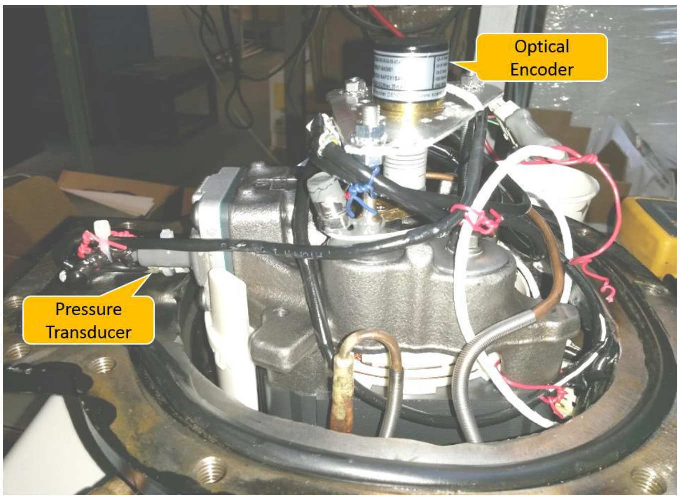

Figure 3. Experimental setup

After the instrumentation of compressor, the pressure measurements were performed while the compressor was running on a fully automated calorimeter system that can maintain the operating conditions at ASHRAE point where the evaporation and condensation temperatures are -23.3°C, 54.4°C, respectively.

5. RESULTS

In the first part, steady state CFD analyzes to determine the flow and force coefficients in accordance to valve lift and piston position were presented. Also, the CFD results which corresponds to the case without piston pin were shared. In the second part, determined coefficients for different piston positions were used in the new 1-D model to predict the capacity, COP and discharge losses. The last part covers the validation of the new 1-D model with the experimental results.

5.1. Flow Coefficients Results

The steady state CFD analyzes were performed for different piston positions. The considered piston positions at corresponding crank angles were shown in Figure 4. The first position illustrates the situation where the discharge valve opens. The flow area is decreasing while the piston pin starts to penetrate into the port cross section. Also, the narrowing gap along with radial flow direction is expected to increase the viscous losses.

IOP Conf. Series: Materials Science and Engineering **604** (2019) 012026 doi:10.1088/1757-899X/604/1/012026

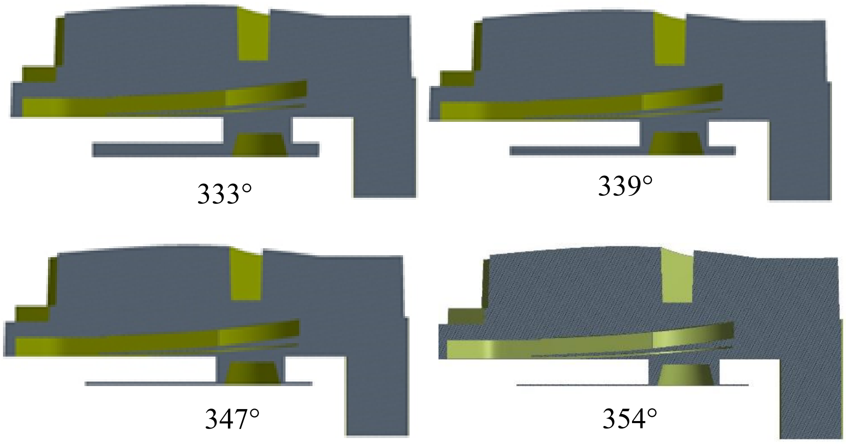

Figure 4. Piston positions at corresponding crank angles

CFD analyzes were performed for different valve lifts at each piston positions. Variation of the velocity vectors on the sectional view with respect to the piston position were shown in Figure 5 for 0.5mm valve lift. For the same valve lift, thanks to the decreasing mass flow rate, the magnitudes of the velocity vectors have decreasing tendency while piston moves to the top dead center (TDC).

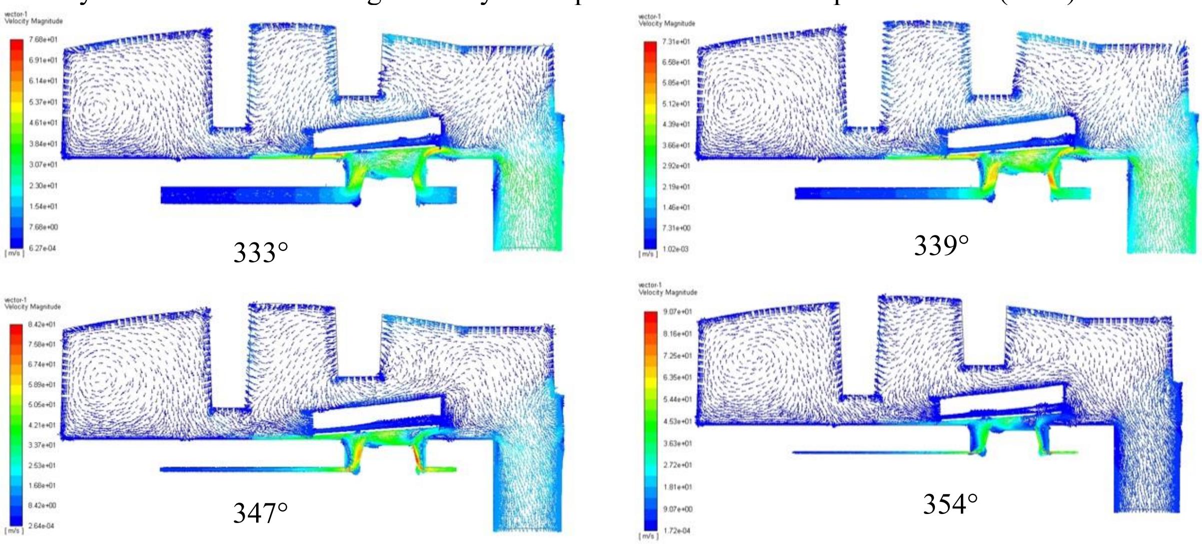

Figure 5. Velocity vectors in different piston positions for 0.5mm valve lift.

Nevertheless, the flow coefficients are decreasing with increasing crank angle as the piston approaches to TDC (Figure 6). Up to 76 % discrepancy was calculated between the lowest and highest crank angels which were taken into account in the calculations. The force coefficients have a similar trend with the flow coefficients. The force coefficients reach to zero value as the piston approaches to TDC.

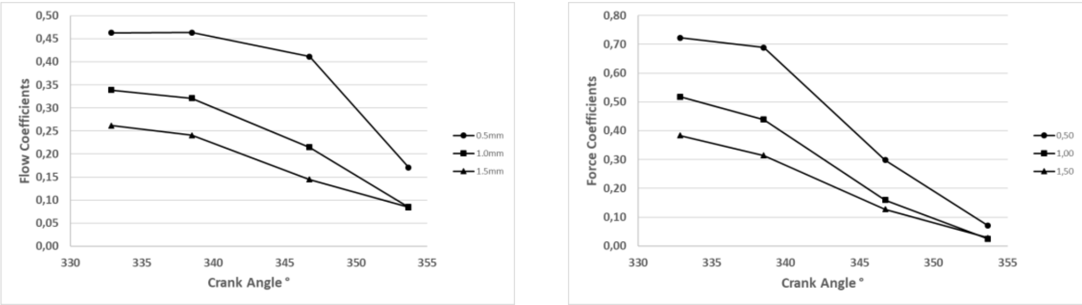

Figure 6. Flow and Force Coefficients

The steady state CFD analyzes were also performed to understand the piston pin effect. The flow and force coefficients for the piston which has no pin were shown in Figure 7. The curves for the flow and force coefficients show similar trend with the cases having piston pin. However, the results showed that flow coefficients have up to 37% higher values without pin case. The differences are diminishing with decreasing valve lift and gap between piston and valve plate.

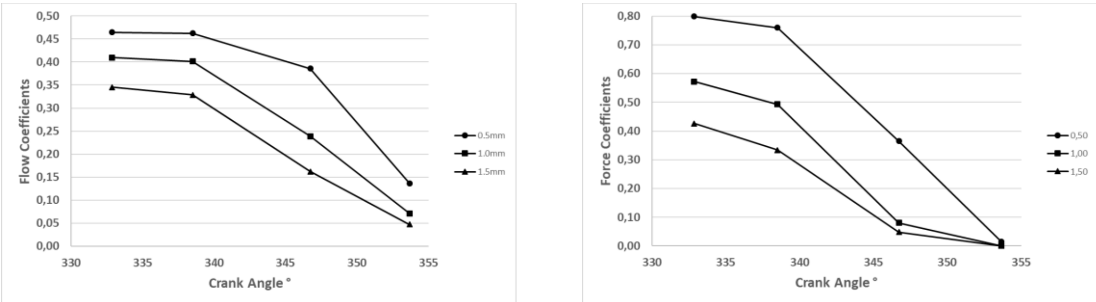

Figure 7. Flow and Force Coefficients (No piston pin)

5.2. Model Results and Experimental Validation

The new 1-D model was used to simulate compressor at the ASHRAE conditions. Table 1 shows the normalized performance results for a compressor speed. In the discharge losses, the area between cylinder pressure in the ideal case (constant pressure) and cylinder pressure in the experiment or model were calculated. The new 1-D model estimates the COP and discharge losses better than previous version which overestimates losses.

Table 1. Performance parameters

ASHRAE	Experimental	1-D Model	New 1-D Model
Capacity	100.0	95.7	104.1
COP	100.0	95.6	99.6
Discharge Losses	100.0	164.3	104.3

Figure 8. shows cylinder pressure values at different normalized rotational speeds. The peak cylinder pressures were estimated better than the previous model. At the beginning of discharge process, calculated mass flow rate is higher than the previous model as the flow and force coefficients are higher. Therefore, cylinder peak pressure reaches to lower level. Also, the rest of the discharge process is calculated with lower flow and force coefficients which provide better estimation of cylinder pressure for the second peak pressure. As a result of better estimation in cylinder pressure during the simulation of the discharge process, the discharge losses were predicted very close to experimental results.

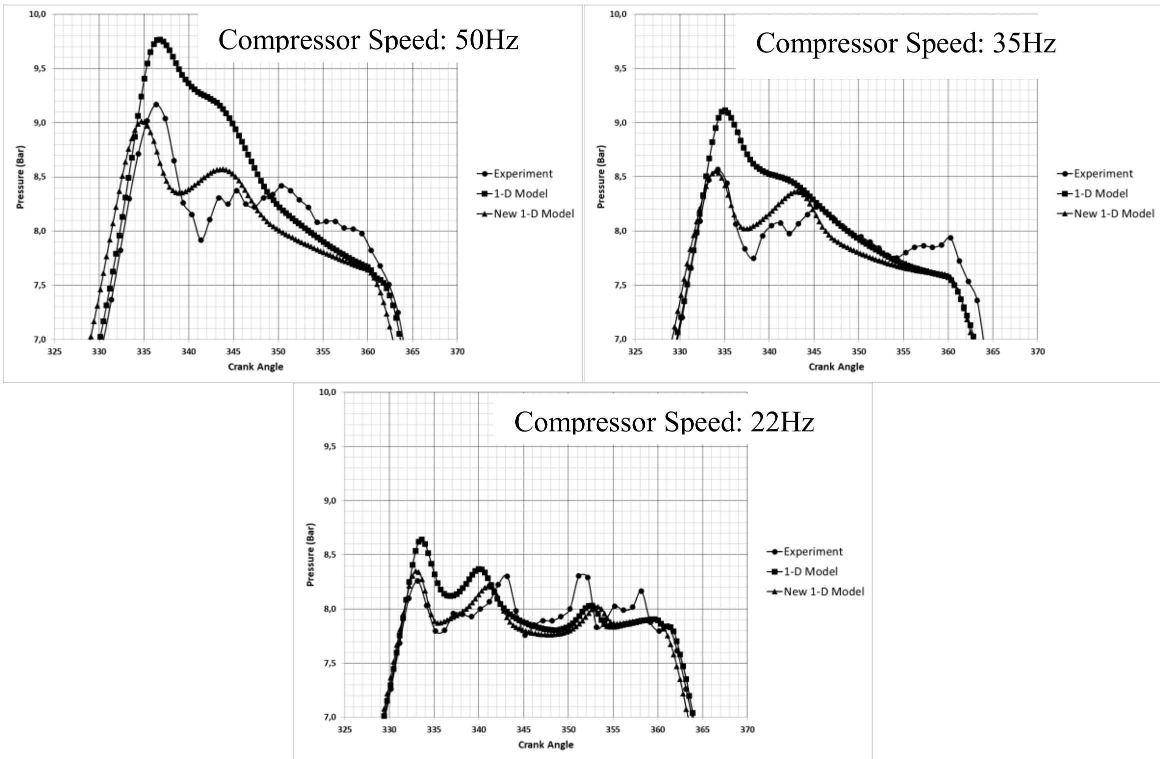

Figure 8. Cylinder Pressure Diagram

6. CONCLUSIONS

In this study, steady state CFD analyzes were preformed to determine flow and force coefficients. The effects of piston position and piston pin were investigated. The determined coefficients for different piston positions and valve lifts were used in the new 1-D model to calculate the performance parameters at ASHRAE conditions. The calculated results were compared with the previous 1-D model and experimental measurements. The followings can be concluded;

- The piston position has a strong effect on flow and force coefficients as the flow area for the radial flow changes with the piston movement. Also, as the piston approaches the TDC, piston pin enters the discharge port which decrease the flow area. According to the steady state CFD analyzes, for the same valve lift, up to 76 % discrepancy was calculated between the lowest and highest crank angels which were taken into account.
- The piston pin which is used to decrease dead volume also influences coefficients. It is shown that flow coefficients have up to 37% higher values without pin case which is expected to decrease peak cylinder pressure and discharge losses.
- Inserting the piston position as a parameter to determine the coefficients in new 1-D model provides better estimation for the cylinder pressure in the discharge process. As a result, discharge process losses were calculated with 4% deviation compared to experimental measurements.

References

[1] Kerpicci H and Oguz E 2006 Transient Modeling of Flows Through Suction Port and Valve Leaves of Hermetic Reciprocating Compressors Int. Compressor Engineering Conf. Paper 1806

[2] Pereira Evandro L L Deschamps C J 2010 A Theoretical Account of the Piston Influence on Effective Flow and Force Areas of Reciprocating Compressor Valves Int. Compressor Engineering Conf. Paper 2010.

[3] LOHN S K, Ernane S, Beppler D L and Deschamps C J 2018 Effect of Fluid Flow Inertia and Backflow on the Effective Force and Flow Areas of Reed-Type Valves Int. Compressor Engineering Conf.
[4] Mu G, Wang F, Mi X and Gao G 2019 Dynamic modeling and analysis of compressor reed valve based on movement characteristics Applied Thermal Engineering 150 (2019) 522–531
[5] Touber S, 1976 A Contribution To The Improvement Of Compressor Valve Design PhD Thesis, Delf University of Technology