2022

Cfd Approach To Optimize Discharge Flow Line Of A Reciprocating Compressor

Sehnaz Ektas

Murat Piri

CFD approach to optimize discharge flow line of a reciprocating compressor

Sehnaz EKTAS[1], Murat PIRI[2]

[1]Arçelik, Compressor Plant,
Eskişehir, Turkey
sehnaz.ektas@arcelik.com

[2]Manisa Uretim ve Teknoloji A.S.,
Manisa, Turkey
murat.piri@arcelik.com

ABSTRACT

The discharge flow line has an important effect on the performance characteristics of a reciprocating compressor. At the beginning stage of a new compressor design, many prototypes should be prepared, and a long testing time is needed in order to decide on the optimum discharge system. In this study, CFD methodology is used to compare different design alternatives of discharge flow line. Appliance of CFD studies in new design projects aims to shorten test time and to reduce the number of prototypes. Differences in design of flow line include change in tube diameters and number of tubes. Mainly, six different alternative configurations are studied. The flow line between the cylinder head and discharge muffler is taken as CFD model geometry. Different mesh sizes are applied in order to verify mesh independency of analyses for all of the scenarios. Mass flow rate and pressure drop along the flow line are the parameters which are compared in order to decide optimum design alternative. Experimental studies are also conducted to verify CFD studies and to determine error rate. The resulting error rate is in an acceptable range so it can be deduced from this study that CFD is a useful tool to optimize such a discharge flow line.

Keywords: Reciprocating compressor, discharge, fluid dynamics, CFD

1. INTRODUCTION

When designing a compressor, heat and mass transfer issues have a high importance. There are two flow lines in a reciprocating compressor, one of them is discharge line and the other one is suction line. Determining temperature and pressure distribution along these lines gives an idea of performance and characteristics of the compressor during the design stage. By making CFD analyses, this determination can be done without any need of prototypes.

In this study, CFD analysis of the discharge flow line is realized for a compressor on its design stage. Six different alternative designs are studied, and their results are compared in order to decide on the optimum one. The prototype of the optimum design is produced at the R&D laboratory. Results of CFD analyses are verified using this prototype.

In the study of Yesilaydin (2018), experimental and numerical analysis are realized to investigate pressure fluctuations at discharge line of a hermetic reciprocating compressor. They used dynamic mesh structure and modeled compression of the piston. Using the pressure difference in the cylinder and discharge manifold, the opening and closing of the discharge valve is modeled. For the experimental study, compressor was instrumented with pressure transducer to obtain the pressure volume diagram.

Kara and Oguz (2010) investigated the temperature distribution of two different crankcase designs both experimentally and numerically in their study. They performed temperature measurements to obtain the gas and solid temperatures. They realized the numerical simulations based on the measured gas temperatures with additional convective heat transfer boundary conditions found in the generally accepted values in the literature.

2. EXPERIMENTAL SETUP

The experiment given in Figure 1 is conducted at the calorimeter system at ASHRAE conditions. While making calorimeter measurements, pressure data on the outlet of the discharge flow line is collected as can be seen in Figure 2. Base design is considered as the protype. The pressure data collected during the test is used to make a comparison between CFD analysis results.

Figure 1: Experimental Setup

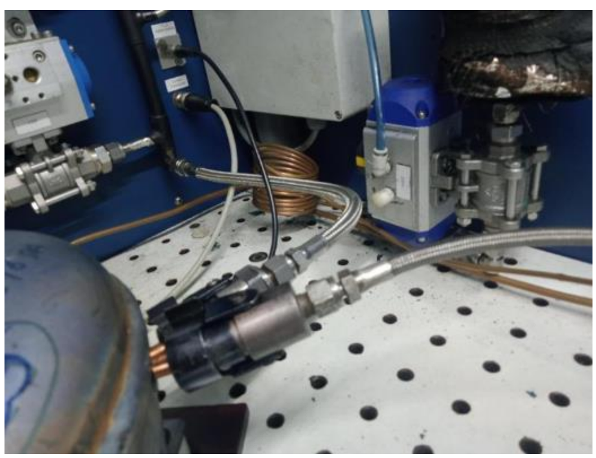

Figure 2: Pressure measurement

3. NUMERICAL STUDIES

Commercially available CFD code ANSYS Fluent is used for these numerical studies. Six different discharge flow line cases are analyzed. Radius and number of connection holes on the crankcase are different from each other for each case. Approximately 9.000.000 elements are used as mesh structure. Mesh structure figures are given in Figure 3 and Figure 4 for Case 1. The same structure is used for all the cases. The fluid domain in the cylinder head, valve plate, discharge mufflers, connection hole and shock loop are used as control volume. Solid domains are not used. Steady state analysis is applied in the study. The fluid properties and the boundary conditions are given in Table 1 and Table 2, respectively. Boundary conditions are the same for all cases. Inlet mass flow rate and temperature are defined as inputs and inlet/outlet pressures and velocities are obtained as outputs. Convection heat transfer is defined at the surface of the model. Energy and flow equations are solved.

Table 1: Properties of fluid

Properties	**Isobutane (R600a)**
Density (kg/m3)	Ideal gas
Specific Heat cp (j/kg-K)	1911
Thermal conductivity (W/m-K)	0.022
Dynamic viscosity (kg/ms)	8.76e-5
Molecular Weight (kg/kgmol)	58.1

Table 2: Boundary conditions

Inlet	Mass flow rate: 0.5 g/s
	Total Temperature: 120 °C
Outlet	Gauge pressure: 0 Pa
	Backflow total temperature: 80 °C
Surface	Convection heat transfer coefficient: 8 W/m-K
	Free Stream Temperature: 80 °C

Study cases are different in terms of their crank case connecting tube hole radius, position and number. Connecting tube hole radii are classified as wide hole, medium hole and narrow hole. The cases are described in Table 3.

Table 3: Study cases

	Case 1	Case 2	Case 3	Case 4	Case 5	Case 6
Number of holes	2	1	1	1	1	2
Position of holes	right/left	right	left	right	right	right/left
Radius of holes	narrow right hole/wide left hole	wide right hole	wide left hole	medium right hole	narrow right hole	narrow right hole/narrow left hole

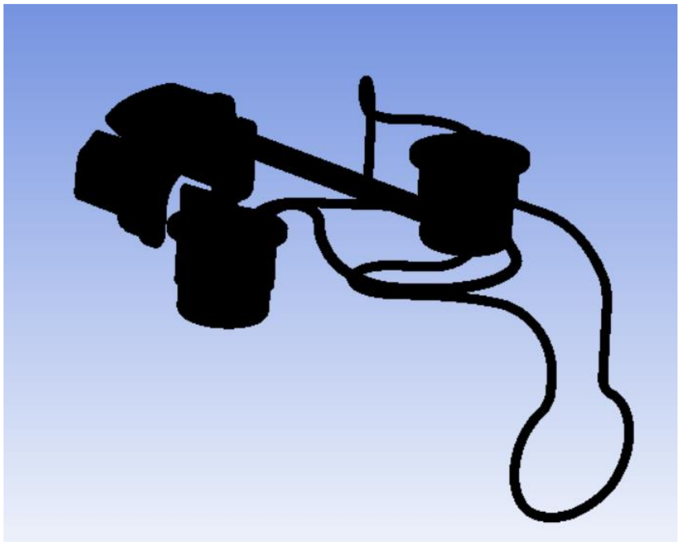

Figure 3: Mesh structure of control volume

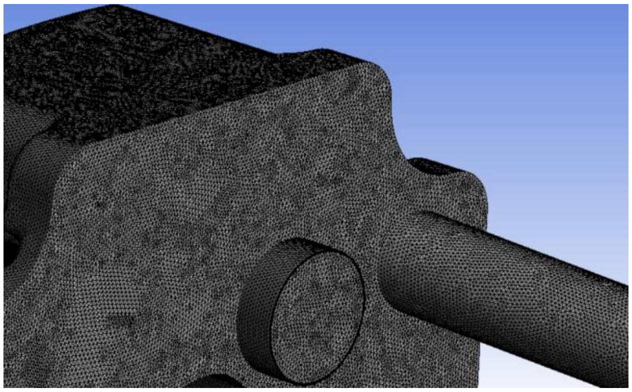

Figure 4: Detail view of mesh structure

4. RESULTS OF NUMERICAL STUDY

As a result of CFD analyses pressure and velocity data at inlet and outlet are obtained. Also, temperature contours are compared. Results are shown in Table 4 and Figure 5.

Table 4: Pressure and velocity at inlet& outlet

	Case 1		Case 2		Case 3		Case 4		Case 5		Case 6	
	inlet	outlet	inlet	outlet	inlet	outlet	inlet	outlet	inlet	outlet	inlet	outlet
Pressure (bar)	0,6	0,01	0,6	0,01	0,64	0,04	0,57	0,02	0,61	0,06	0,61	0,02
Velocity (m/s)	3,39	70,30	3,38	70,90	3,30	67,10	3,44	68,08	3,37	70,68	3,37	70,65
Pressure difference between inlet & outlet (bar)	0,59		0,59		0,60		0,55		0,55		0,59	
Velocity difference between inlet & outlet (m/s)	66,89		67,49		63,83		64,64		67,31		67,28	

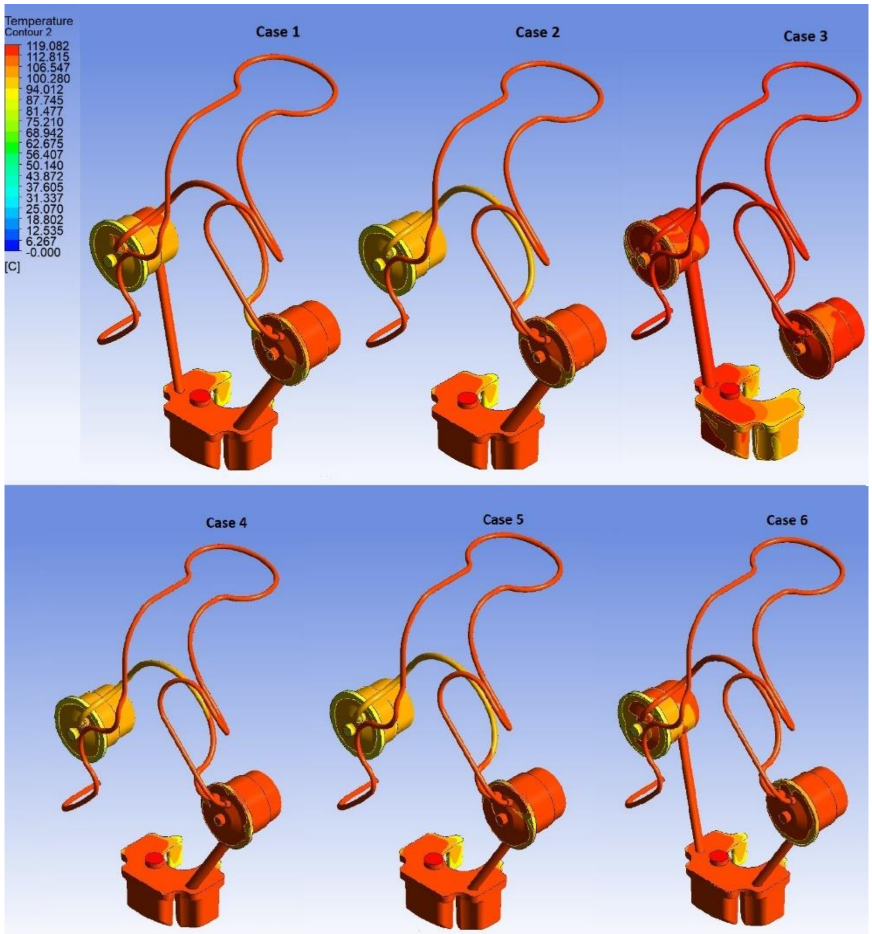

Figure 5: Temperature contour of different cases

Case 3 has the minimum velocity difference at inlet and outlet, and it has the maximum pressure drop. This situation is considered as optimum case because pressure drop is aimed along the discharge valve and minimum velocity difference is preferred for noise issues. Case 3 is used as the protype model for experimental study. Experimental pressure is measured as 0,63 bar at the outlet. This corresponds to a %1 difference between numerical and experimental studies.

5. DISCUSSION AND CONCLUSION

This paper presents CFD analyses of different discharge flow line configurations for a reciprocating compressor. The results of CFD analysis is verified by an experimental study for one of the configurations. This verification has shown that the CFD analysis can be used as a reliable tool to make optimization for designing discharge flow line. As a further study, suction flow line can be also optimized by using the same analysis method.

NOMENCLATURE

CFD	Computational Fluid Dynamics

REFERENCES

Yesilaydin, I., Erbay, L.B., (2018). A Study on a Numerical Modelling of Discharge Line Flow Analysis of a Household Type Refrigerator, International Compressor Engineering Conference, Purdue University, West Lafayette, USA.

Kara, S., Oguz, E. (2010). Thermal Analysis of a Small Hermetic Reciprocating Compressor, International Compressor Engineering Conference, Purdue University, West Lafayette, USA.

ACKNOWLEDGEMENT

We would like to extend deep gratitude our entire Arcelik Company for their support in writing this paper.