Numerical Simulation of Fluid Flow through Reciprocating Air Compressor Valve

[1]Vedant Panchal, [2]Pravin Zinzala,
[1]M.E. Scholar, [2]Assistant Professor,
[1]mechanical Department of Post Graduation Section,
[1]L.J. Institute of Engineering Technology, Ahmedabad, Gujarat, India

Abstract: *It is necessary to understand the behavior of the gas flows in the compression chamber generated by the driven piston, and the behavior of the compressor valve that are opened and shut by the fluid force, when we reduce the loss of suction and discharge of the reciprocating compressor. To understand the above phenomenon, the moving object such as piston and reed valves is analyzed by applying Fluid structure interaction (FSI) method that is suitable for expressing complex shape as main grid in this study. In this paper, first, we introduce our numerical analysis method.*

Keyword: Fluid Flow, Reciprocating Compressor, CFD, FSI

I. Introduction

Compressor is widely used in many industries now days. Compressor work on old principle of the piston-cylinder mechanism. In recent years, efforts are energy saving have been work all over the world from the perspective of conservation of the global environment. Including the Asian countries, where economic growth is marked and the European countries, which are working positively on energy efficient, energy efficient regulations, is becoming important year by year.

This trend has accelerated the energy saving competition among refrigerator manufacturers in global market and it will be necessary to developing higher efficiency reciprocating compressors in next coming year. The behavior of refrigerant in the compressor is analyzed by CFD gave a idea on the design, to minimize the experiments cost and to accelerate the development. Reciprocating compressors are also used in the many industries now days. Compressed are used in automobiles, hospitals, jet and other appliances. Recently, it begins to be used FSI (Fluid Structure Interaction) analysis that considers the behavior of suction valve and discharge valve. However, a general method of FSI analysis has some demerit that the mesh around the object is transformed and deteriorated. Then, it introduces following analysis method to solve the above problem. The three-dimensional compressible Navier - Stokes equations are used as the governing equations.
For developing a compressor a sizing of the compressor are important.[1] piston displacement are give the most vital role in the compressor sizing. Two mayor aspects were analyzed and found to drastically change in shape and magnitude. Pressure profiles over the valve disc change from a nearly to p_h at distribution to a smoother bell-shaped profile with the change of geometry.[2] The virtual flux method has been developed for simulating fluid-structure interaction problems. The numerical flux across the fluid-structure interface is

successfully replaced with the virtual flux, so that proper interface conditions are satisfied there.[3] Using the FEA the designer can calculate the centerline diameters of the rings and seats for machining. In operation, with thermal expansion, good response of the valve is obtained and adequate tightness is achieved.[4] The average temperature over time of the suction muffler outlet is thought to be pulled up by heat transfer from the cylinder block and the valve plate when the suction lead is closed.[5]

II. Reciprocating Compressor

Reciprocating compressors are the best known and most widely used compressors of the positive displacement type. They operate on the same principle as the old, familiar bicycle pump mechanism, by means of a piston in a cylinder. As the piston moves forward in the cylinder, it compresses the fluid into a smaller space, thus pressure rising. The basic reciprocating compression element is a single cylinder compressing on one side of the piston (single-acting). A unit compressing on both sides of the piston (double-acting) consists of two basic single-acting elements operating in parallel in one cylinder. Most of the compressors in use are of the double-acting compressor.

III. Piston Displacement

The piston displacement is the total volume actually displaced by the compressor piston at given speed, as the piston travels the length of its stroke from bottom dead center to top dead center. The full stroke and thus the piston displacement are represented by the travel of the piston from TDC to BDC. This volume is usually expressed in CFM (cubic feet per minute). For multistage units, the piston displacement of the first stage alone is commonly stated as that of the full mechanism. In the case of a double-acting cylinder, the displacement of the crank end of the cylinder is also included. The crank end displacement is less than the head end displacement by the amount that the piston rod displacement. The piston displacement (PD) for a single-acting unit is readily computed by the following formulas:

Calculating PD for a single-acting cylinder:
PD = AHE x S\12 x rpm………………………………………………………………………..… (i)

Where,
AHE = area of head end of piston in square feet
S = stroke in inches
RPM = revolutions per minute
PD = piston D is displacement in cubic feet per minute

IV. Parts of Compressor

Compressor has many parts to perform the work, but there some important parts given below

- ✓ Piston
- ✓ Crankshaft
- ✓ Cylinder
- ✓ Valve
- ✓ Intercooler
- ✓ Crankcase
- ✓

Importance of valve in compressor

In this project we will talk about how valve effect the performance of reciprocating compressor and valve gave major role of the compressor failed. Mostly of the 36% of compressor failed because of the valve in survey.

A reciprocating or piston compressor is a compressor that is piston-driven by a crankshaft in order to deliver high-pressure gas. The compressor captures a volume of gas from a suction port and transfers it into a cylinder, where it is trapped and compressed by a piston that reduces its volume. Thereafter, the compressed gas is discharged through the exhaust port into the discharge pipe. The flow of the gas through the cylinder is controlled by valves.

If the corrosive elements are present in the compressed gas, they might bring about the corrosion of the valve parts. This is especially dangerous to the springs as they can fail prematurely due to the corrosion-related fatigue. However, if the gas composition is known during the design process, suitable corrosive resistant materials can be chosen and therefore the effects of the corrosion can be limited. The application conditions refer mostly to the quality of the compressed gas. The presence of dirt or debris will accelerate wear, can limit opening and closing of the valve, and in extreme conditions, the valves can become blocked

V. Valves

Compressor valves are devices placed in the cylinder to permit one-way flow of gas either into or out of the cylinder. There must be one or more valves for inlet and discharge in each compression chamber (cylinder end). The reciprocating compressor uses automatic spring-loaded valves that open only when the proper differential pressure exists across the valve. Inlet valves open when the pressure in the cylinder is slightly below the intake pressure. Discharge valves open when the pressure in the cylinder is slightly above the discharge pressure.

V.I. Function of a Compressor Valve

A compressor valve regulates the cycle of operation in a compressor cylinder. Automatic compressor valves are pressure activated, and their normal movement is controlled by the compression cycle. The valves are opened solely by the difference in pressure across the valve; no mechanical device is used. The best illustration of a compressor valve cycle is obtained by correlating the piston movement to the pressure volume diagram.

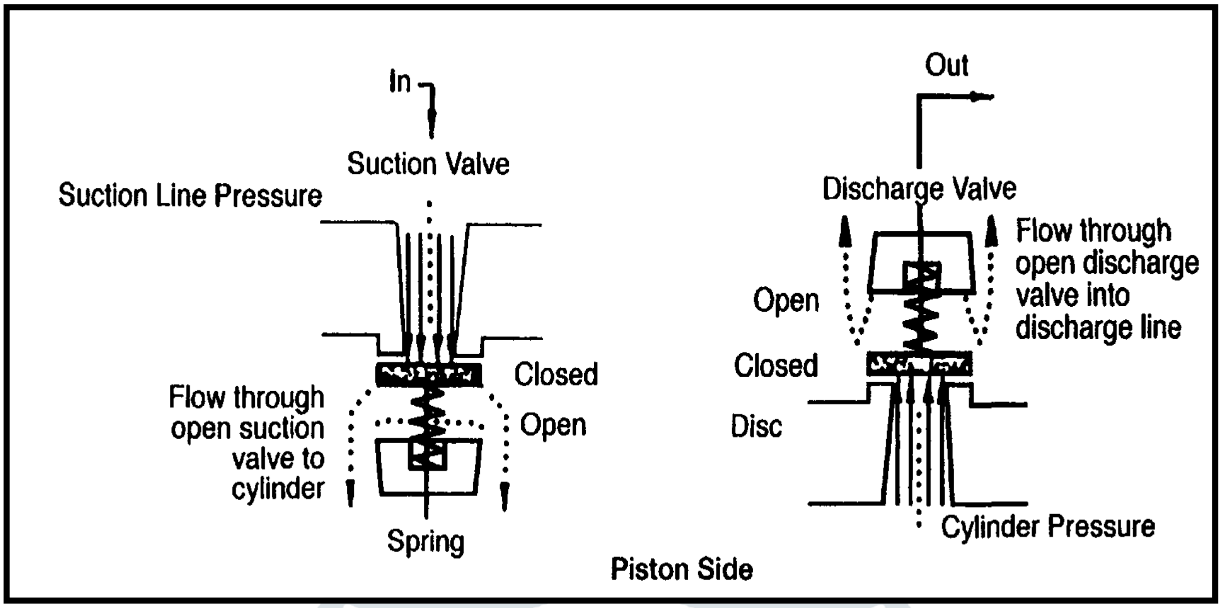

Fig.1 function of valve

VI. APPLICATION

VI.I. Numerical scheme

The numerical scheme of FLUENT solver is based on finite volume technique and involves following steps: Integration of the governing equations of fluid flow over all the control volumes of the solution domain. The terms in the integrated equation representing flow process such as convection, diffusion and source with finite difference type approximation and thereby converting them into algebraic equations. Solution of the algebraic equations is obtained by an iterative method

VI.II. Equations solved by the solver

The equations which were enabled to be solved by the solver were as below:

• Energy

• Mass and Momentum

• Turbulence

Discritization scheme

a) Pressure Interpolation Scheme: Standard

b) Pressure–velocity coupling: SIMPLE (Semi-Implicit Method for Pressure Linked Equation) was used. The Simple Algorithm uses a relationship between velocity and pressure.

c) Momentum, Turbulent Kinetic Energy and Turbulent Dissipation rate, Energy Equation: 1st order upwind scheme was used.

VI.III. Viscous model

Standard *k*-ε model was used in these simulations. The simplest "complete model" of turbulence are two-equation models in which the solution of two separate transport equations allow the turbulence velocity and length scale to be independently determined. The standard *k*-ε model in FLUENT falls within this class of turbulence model and has become the favorite choice of practical engineering flow calculations in the time. Robustness, economy, and reasonable accuracy for a wide range of turbulent flows explain its popularity in industrial flow and heat transfer simulations. It is a semi-empirical model and the derivation of the model equation relies on phenomenological consideration and empiricism.

VI.IV. Dynamic mesh strategy

In an engine cylinder, transient simulation is possible only with a deforming computational grid. With the dynamic mesh capability of FLUENT the geometry needs to be meshed at one defined crank angle. From here on the solver will do all necessary mesh modifications? Node and cell movement as well as the internal structure of the mesh are a result of the boundary conditions defined for the moving parts. The mesh update can be achieved by spring motion, local re-meshing, dynamic layering and a combination of these.

VI.V. Sliding mesh strategy

When a time-accurate solution for moving mesh is desired, the sliding mesh model is used to compute the unsteady flow field. The sliding mesh model is the most accurate method for simulating flows in multiple moving reference frames, but also the most computationally demanding. Most often, the unsteady solution that is sought in a sliding mesh simulation is time periodic. That is, the unsteady solution repeats with a period related to the speeds of the moving domains. However, we can model other types of transients, including translating sliding mesh zones.

VI.VI. Boundary conditions

The flow domain considered for simulation is downstream of the Compressor. Inlet valve movement is simulated using the ideal valve lift profile of the engine. The intake valve open at TDC and the boundary condition of the inlet is assigned as the pressure inlet. The inlet valve is closed after 180 degree.

VI.VII. Computational Procedure

Computations have been made for operational speed ranging from 1000 rev/min and time step of the order of 1 degree Crank Angle. The calculations commence with the piston at the TDC (90 degree CA), with inlet valve and exhaust port/valve closed. Using the initial and boundary conditions as mentioned earlier, the computation proceeds with the piston descending downwards and the inlet valve begin to open so as to allow fluid to enter the flow domain i.e. cylinder. The opening and the closing of the inlet valve take place according to the valve timing (ideal or actual) of the engine. The inlet valve closes at BDC 180 CA at which the boundary condition of the inlet is changed from pressure inlet to wall to prevent fluid escaping from the cylinder.

VI.VIII. Assumptions

The gas is assumed as an ideal gas the density is calculated using the ideal gas law. The ideal gas law states that molecules do not attract each other, and the molecules themselves do not take up volume. This is problematic if the pressure of the gas is maybe hundred times greater than atmospheric pressure or at very low temperatures. The operating pressure in the compressor is set to 2.03 bar and the temperature of the gas in the suction pipeline is set to 323.15 K, which is acceptable for ideal gas calculations. The CFD/FSI model it is assumed the piston moves at a constant angular velocity.

VI.IX. Model of the valve in compressor

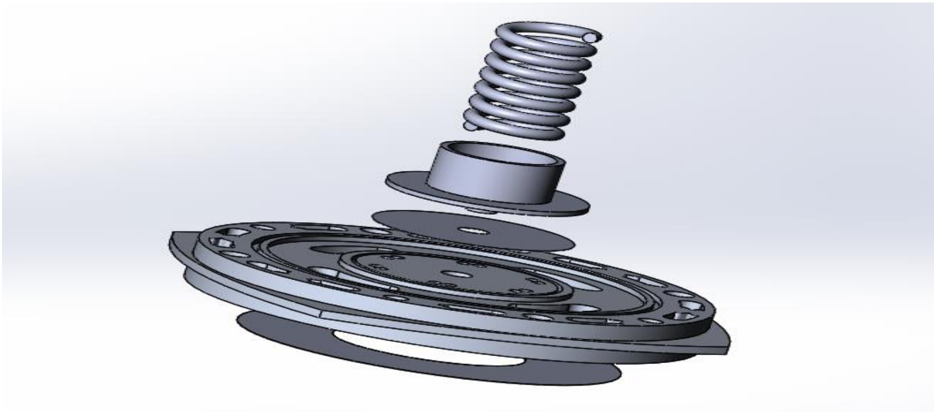

Fig.2 Ring valve

VII. CFD/FSI Model Flow through the valve

The flow through the valves is considered as (quasi stationary) outflow of gas from a pressurized vessel through a convergent nozzle. In case of constant heat capacities it is given by St.Venant and Wantzell

$$\frac{dm}{dt} = \alpha p_s A \sqrt{\frac{2\kappa}{(\kappa-1)\bar{R}T_s}} \sqrt{r^{2/\kappa} - r^{(\kappa+1)/\kappa}} \quad \text{(ii)}$$

And

$$r = \max\left[\frac{p_c}{p_s}, \left(\frac{2}{\kappa+1}\right)^{\kappa/(\kappa+1)}\right] \quad \text{(iii)}$$

Where,
Ps- pressure in the suction chamber,
T s- temperature in the suction chamber,
K- Adiabatic exponent in the valve conditions,
R- Equivalent gas constant in the valve conditions,
Pc- pressure in the cylinder,
α- valve flow coefficient,
A- Suction valve flow area,
Dm/dt: mass rate of flow through the valve

In this section the methods used when setting up the 3-D CFD/FSI model is presented. The geometry of the simple compressor is introduced. A dynamic mesh method accounting for the movement of the valve and the piston is introduced and how to use the method is explained. This method includes a "Smoother", a "Remesher", and a "Layering" option. An event mode is also used when setting up the model. Compressible flow is used for the simulations since pressure deference is necessary to simulate the fluid structure interaction on the suction valve. Fluid structure interaction is an event where a fluid creates enough force on an object/structure to have the object/structure move or deform/skew. For this simulation only the movement of the structure is of interest and therefore no stress or strain analysis have been performed. To calculate compressible flows the energy equation is enabled. The ideal gas law is used to calculate the density of the gas.

VIII. Canclution

We estimated the pressure changes on time in the cylinder following consideration of the frequency of vibration of the suction lead valve using the pressure difference and made that pressure in the cylinder the boundary condition to carry out three-dimensional CFD of the suction muffler. FLUENT is used for CFD. After the using of the ansys work beach for the solving the compressorr valv phenonomia by the FSI (fluid strucutre intraction) that mode of the system used to sovle both structure and fluent at a two way coupling scheam.it easy way to indentified the problem of the pressure drop and volumetric efficiency of the compressor valve.

IX. Acknowledgment

We take this opportunity to express our gratitude to our loving parents who are the center our inspiration

References

1. Design Calculations to Evaluate Performance Parameters of Compressor Valve,Chaitanya D. Patil1, Prof. M. R. Khodke, AkashBaheti,International Journal of Advance Engineering and Research Development Volume 4, Issue 7, July
2. Numerical analysis of the turbulent fluid flow through valves. Geometrical aspects influence at different positions'.J. Rigola, D. Aljure, O. Lehmkuhl, C.D. P´erez-Segarraand A. Oliva,International Conference on Compressors and their Systems-2015
3. Study of CFD Considering Valve Behavior in Reciprocating Compressor,Hikichi, Takumi; and Morinishi, Koji, Kinjo, Kenji; Nakano, Akira,*International Compressor Engineering Conference. Paper 1975, 2010*
4. Design of Valves Used in Reciprocating Compressors,MirceaHoriaTierean, Liana SandaBaltes, Advances in Applied and Theoretical Mechanics
5. CFD applications for Development of Reciprocating Compressor,Akira Nakano, Kenji Kinjo,*International Compressor Engineering Conference. Paper 1842, 2008*