2004

Analysis of Refrigerator Compressor Using CFD

V. Koterswara Rao  
*Applicomp (India) Limited*

Mahesh S. Murthy  
*Applicomp (India) Limited*

S. Armuga Raja  
*Applicomp (India) Limited*

Dattu Kumar  
*Applicomp (India) Limited*

Follow this and additional works at: https://docs.lib.purdue.edu/icec

https://docs.lib.purdue.edu/icec/1695
ANALYSIS OF REFRIGERATOR COMPRESSOR USING CFD

V. Koteswara Rao, Mahesh S. Murthy, S. Arumuga Raja & Dattu Kumar
Applicomp India Limited, R&D, Bangalore, Karnataka, India
Ph: +91 80 27821845/46, Fax: +91 80 27821154, E-mail: kotivkr@applicompindia.com

ABSTRACT

In this paper we present a case study wherein CFD (Computational Fluid Dynamics) technique was used to study the suction gas flow through the suction muffler for improving the efficiency and reduce the noise level of a hermetic refrigerator compressor.

1. INTRODUCTION

The growing demand for efficient and silent Compressor coupled with intense competition in the market has lead to frequent change in design of product. Such changes have to be ensured for its reliability and the time frame does not permit much of iteration to be carried out. Hence there is a need to use relevant technique to simulate and predict the outcome of various design parameter change.

The most important factor, which affects the cooling capacity of a compressor, is volumetric efficiency. The factors affecting volumetric efficiency are suction gas superheating in the suction path, throttling, clearance volume, valve dynamics, blow back of gases through clearance in piston and cylinder and heat transfer during its travel into the cylinder. In this paper two factors were considered for volumetric efficiency improvement one is suction gas superheating and the other is restriction of passage for gas flow. This is a very complex phenomenon, which requires a great deal of time and money to study and understand. To understand the refrigerant behaviour through suction muffler CFD was used in analyzing these complex phenomena to overcome the time constraint otherwise involved.

2. CFD MODELLING AND BOUNDARY CONDITIONS

CFD in one form or other is basically governed by fundamental equation of fluid dynamics – the Continuity, Momentum and Energy Equation. The law which constitute the Equation are

1. Law of conservation of mass
2. Newton’s Second law \( F = m \times a \)
3. Law of conservation of energy

The CFD analysis involves the following steps

2.1 Modeling of flow path

Initially the components through which the gas flow path to be studied were modeled using Pro-E software. This model was then converted in to IGES (Initial Graphic Exchange Specification) format. The IGES format was then imported in ANSYS / FLOTRAN (CFD). Here the model was inverted so that only the gas flow paths exist as model. This model was subjected to geometric clean up, sharp corner and radius were eliminated to improve the node connectivity after meshing.
2.2 Meshing
Meshing or Discretizing is a process where in the solid modeling is divided into a number of small elements. In this analysis Fluid 142 element was used in meshing.

2.3 Applying the boundary, load conditions and material properties
In this Analysis the velocity vector of refrigerant in contact with surface of the component was considered as zero (simulating a practical condition). The inlet pressure and temperature were varied till the required mass flow was matching to experimental value in case 1. Now this pressure and temperature was considered as the input in the second case with suction separator in the cylinder head. The material and fluid properties include density, viscosity, specific heat, and thermal conductivity for solving the governing equation of CFD.

2.4 Solving the solution
Here the software performs several number of iteration to solve the problem.

2.5 Post processing the results
Post processing the result is the output of CFD analysis, the output are
1. Pressure plot
2. Temperature distribution
3. Velocity of flow.

3. ANALYSIS
The analysis is carried out on hermetic refrigerator compressor used with R134a refrigerant fitted in direct cooled and frost free refrigerator. In this model there are two types of suction muffler. The first design (Design 1) is having Low noise (noise level of <40dBA) and energy efficiency ratio (EER) of 3.6. The second design (Design 2) is having normal noise (noise level of <45 dBA) and better energy efficiency ratio (EER) of 3.8. The basic objective was to design a new muffler that is efficient (Design 2) and silent (Design 1).

The new suction muffler was then analyzed using the CFD technique to finalize the design in single iteration which otherwise would require more time in developing the tool, component and testing.

The CFD analysis for suction flow path with design 3 suction muffler was done in two cases.

1. With suction separator (Design 3 (1))
2. Without suction separator (Design 3 (2))

In case 1 i.e without Suction separator the pressure and temperature at the exit of Suction muffler is high as compared to case 2 with suction separator. Here the reason can be attributed to super heating of gases in the cylinder head.

In case2 with increase in velocity of the refrigerant mass flow increased there by enhancing the output of compressor. Further the EER (Energy efficiency ratio) is also increased. The result thus obtained in CFD Analysis are tabulated below
Table 1: Comparison of results of CFD analysis for design 3

<table>
<thead>
<tr>
<th></th>
<th>WITH OUT SUCTION SEPARATOR (Design 3(1))</th>
<th>WITH SUCTION SEPARATOR (Design 3(2))</th>
</tr>
</thead>
<tbody>
<tr>
<td>MASS FLOW</td>
<td>32.33 gr/min</td>
<td>32.76 gr/min</td>
</tr>
<tr>
<td>INLET VELOCITY</td>
<td>4.37 m/Sec</td>
<td>4.64 m/Sec</td>
</tr>
<tr>
<td>OUTLET VELOCITY</td>
<td>5.86 m/Sec</td>
<td>5.94 m/Sec</td>
</tr>
<tr>
<td>INLET PRESSURE (Kg/cm²)</td>
<td>0.137</td>
<td>0.137</td>
</tr>
<tr>
<td>OUTLET PRESSURE(Kg/cm²)</td>
<td>0.087</td>
<td>0.0801</td>
</tr>
<tr>
<td>INLET TEMPERATURE (K)</td>
<td>305.9</td>
<td>305.9</td>
</tr>
<tr>
<td>OUTLET TEMPERATURE (K)</td>
<td>305.9</td>
<td>304.9</td>
</tr>
<tr>
<td>MACH NUMBER (INLET)</td>
<td>0.013</td>
<td>0.014</td>
</tr>
<tr>
<td>MACH NUMBER (OUTLET)</td>
<td>0.017</td>
<td>0.018</td>
</tr>
<tr>
<td>CAPACITY (CALCULATED)</td>
<td>344.59 BTU/Hr</td>
<td>349.36 BTU/Hr</td>
</tr>
</tbody>
</table>

From the velocity the mass flow was calculated

![Diagram of designs](image-url)
VELOCITY PLOT FOR DESIGN 3(1)

Average Velocity at inlet
\( (V_i) = 4.4 \text{ m/sec} \)

Average Velocity at outlet = 
\( (V_o) 5.7 \text{ m/sec} \)

Vector plot at inlet

Vector plot at outlet
VECTOR PLOT FOR DESIGN 3(2)

Average Velocity at inlet
\( (V_i) = 4.64 \text{ m/sec} \)

Average Velocity at outlet
\( (V_o) = 5.94 \text{ m/sec} \)
4. EXPERIMENTAL SET UP

Once after establishing in CFD that there is an increase in flow rate of refrigerant with the use of suction separator inside the cylinder head, the suction separator component was developed. The developed suction separator was used and compressor was assembled. It was then subjected to test on a secondary calorimeter at standard ASHRAE Condition. Thermocouples were inserted at appropriate point to obtain temperature at various points in suction path.

The result thus obtained in Experimental set up are tabulated below

5. EXPERIMENTAL VALUE

Table 2 Comparison of Experimental values

<table>
<thead>
<tr>
<th></th>
<th>WITHOUT SUCTION SEPARATOR</th>
<th>WITH SUCTION SEPARATOR</th>
</tr>
</thead>
<tbody>
<tr>
<td>MASS FLOW</td>
<td>32.3 gr/min</td>
<td>33.2 gr/min</td>
</tr>
<tr>
<td>CAPACITY</td>
<td>344.15 BTU/Hr</td>
<td>353.72 BTU/Hr</td>
</tr>
</tbody>
</table>

The capacity increase for same noise level with new suction muffler and separator is 2.5% ~ 3% and efficiency increase is 4 ~ 5%. We have also redesigned the motor and added run capacitor for increasing the overall efficiency from 3.6 to 4.0 BTU/W.Hr. The details of various test carried out are as follows.

Result comparison of various designs / Model is tabulated:

Table 3 Comparison of Results for 4.17 cc and 6.5 cc

<table>
<thead>
<tr>
<th>MODEL</th>
<th>DESIGN 1</th>
<th>DESIGN 2</th>
<th>DESIGN 3(1)</th>
<th>DESIGN 3(2)</th>
</tr>
</thead>
<tbody>
<tr>
<td>4.17 C.C</td>
<td>CAPACITY (BTU/Hr)</td>
<td>340</td>
<td>353</td>
<td>347</td>
</tr>
<tr>
<td></td>
<td>EER (BTU/Hr / W hr)</td>
<td>3.6</td>
<td>4.0</td>
<td>3.96</td>
</tr>
<tr>
<td></td>
<td>NOISE LEVEL</td>
<td>&lt;40 dBA</td>
<td>45 dBA</td>
<td>&lt;40 dBA</td>
</tr>
<tr>
<td>6.5 C.C</td>
<td>CAPACITY (BTU/Hr)</td>
<td>516</td>
<td>557</td>
<td>548</td>
</tr>
<tr>
<td></td>
<td>EER (BTU/Hr / W hr)</td>
<td>3.65</td>
<td>3.97</td>
<td>3.94</td>
</tr>
<tr>
<td></td>
<td>NOISE LEVEL</td>
<td>&lt;42 dBA</td>
<td>47 dBA</td>
<td>&lt;42 dBA</td>
</tr>
</tbody>
</table>

CONCLUSION

The Computational fluid dynamics can be used as a tool in evaluating various parameter like temperature, pressure, velocity at the required area of interest with out the actual physical component by simulating the test similar to the practical condition.

CFD technique helps in evaluating the results with various change in design parameters in less time, which other wise would require more time in developing prototypes and testing. The test result comparison between experimental and CFD are almost matching which shows that they are reliable and fast method of evaluating the test.
ACKNOWLEDGMENT

The authors take this opportunity to express their thanks to M/s Haritha Infoserve who supported them in accomplishing the project.

REFERENCES

1. James R. Lenz, Edward A. Cooksey, 2002 “Application of Fluid Dynamics to Compressor Efficiency Improvement” International Compressor Engineering Conference at Purdue; 441,446