

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>

Rao, V. Koterswara; Murthy, Mahesh S.; Raja, S. Armuga; and Kumar, Dattu, "Analysis of Refrigerator Compressor Using CFD" (2004). *International Compressor Engineering Conference*. Paper 1695.
<https://docs.lib.purdue.edu/icec/1695>

This document has been made available through Purdue e-Pubs, a service of the Purdue University Libraries. Please contact epubs@purdue.edu for additional information.

Complete proceedings may be acquired in print and on CD-ROM directly from the Ray W. Herrick Laboratories at <https://engineering.purdue.edu/Herrick/Events/orderlit.html>

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))

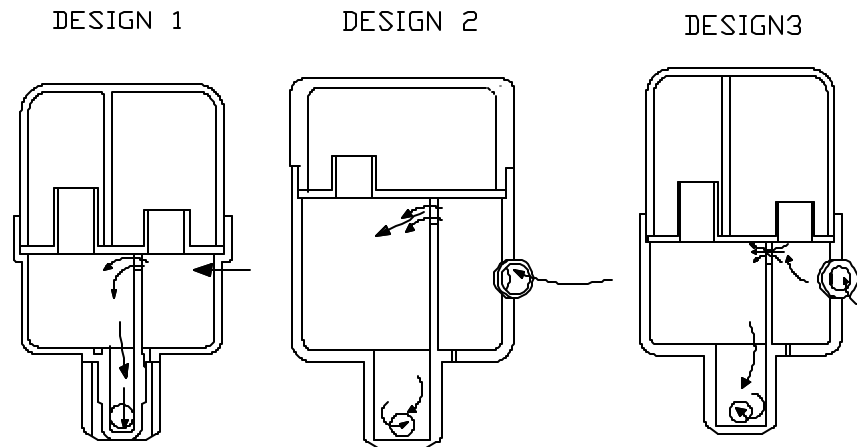
In case 1 i.e without Suction separator the pressure and temperature at the exit of Suction muffler is high as compared to case 2 with suction separator. Here the reason can be attributed to super heating of gases in the cylinder head

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

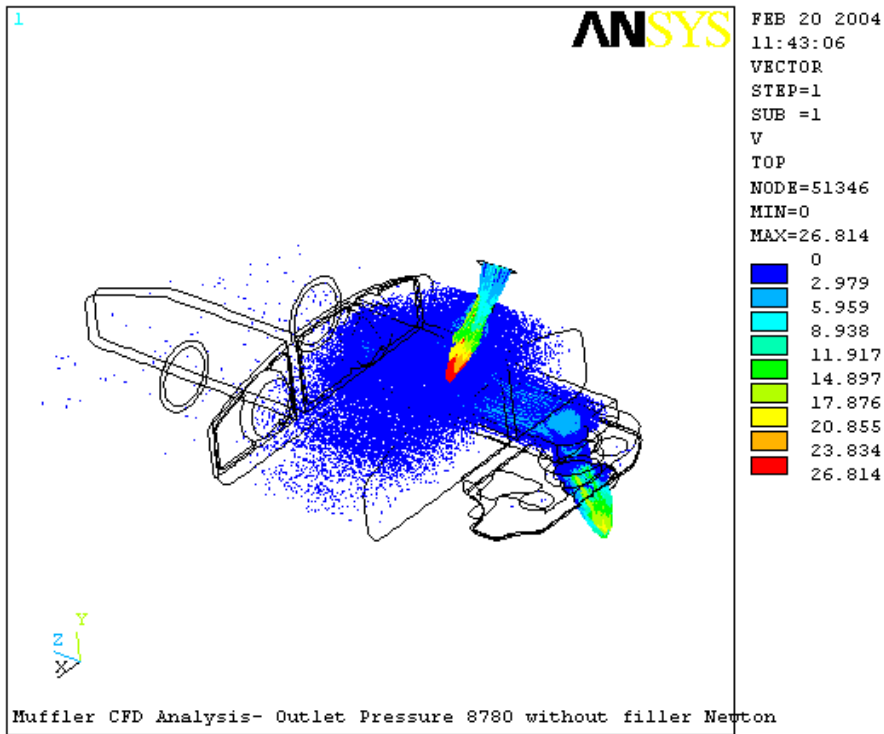
Table1 Comparison of results of CFD analysis for design 3

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

From the velocity the mass flow was calculated

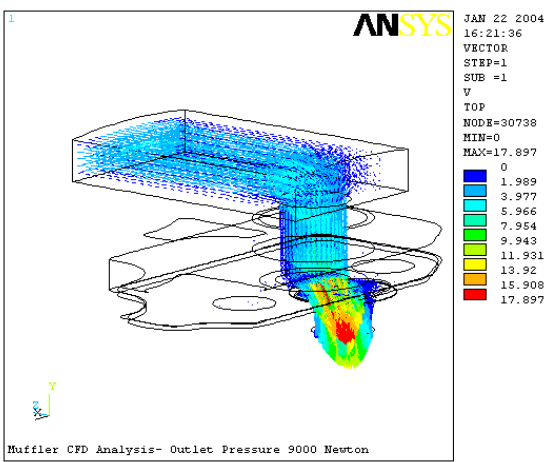
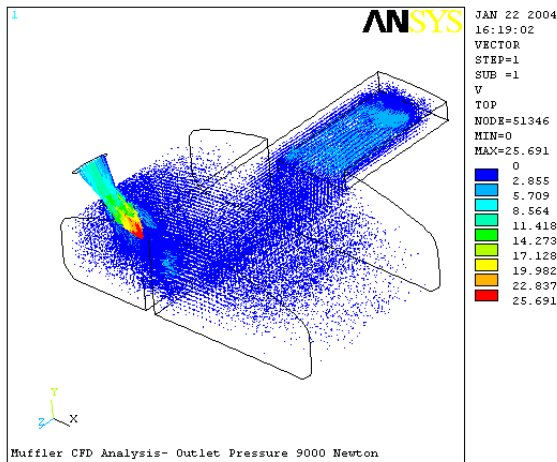


VELOCITY PLOT FOR DESIGN 3(1)



**Average Velocity at inlet
 (V_i) = 4.4 m/sec**

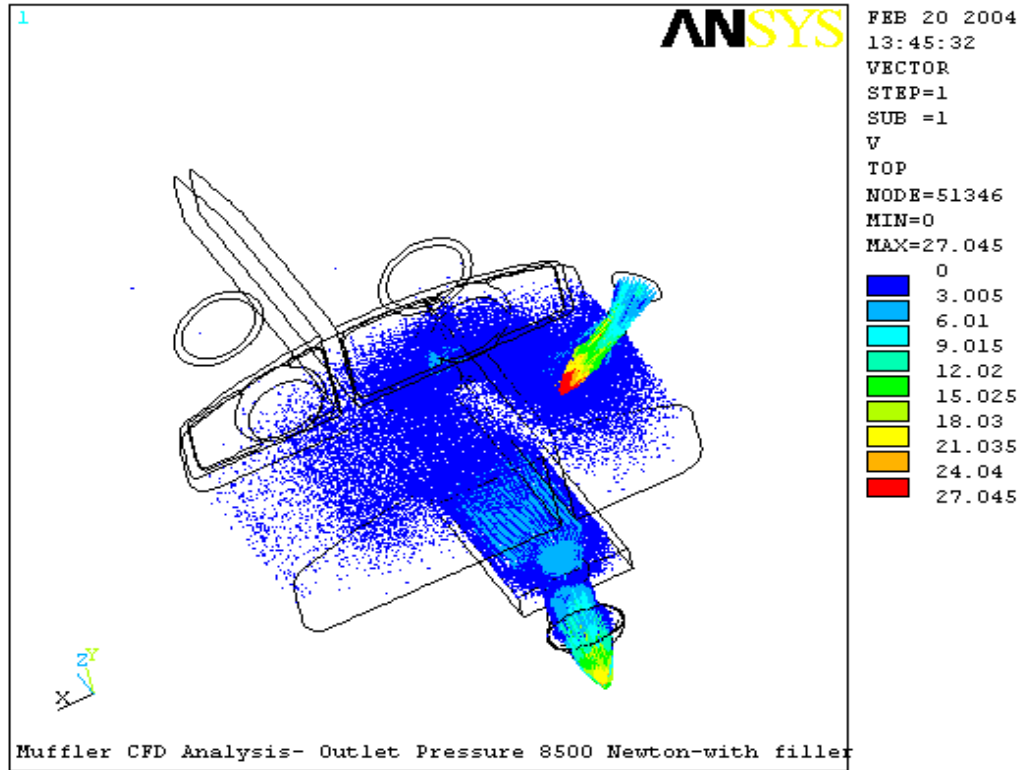
**Average Velocity at outlet =
 (V_o) 5.7 m/sec**



Vector plot at inlet

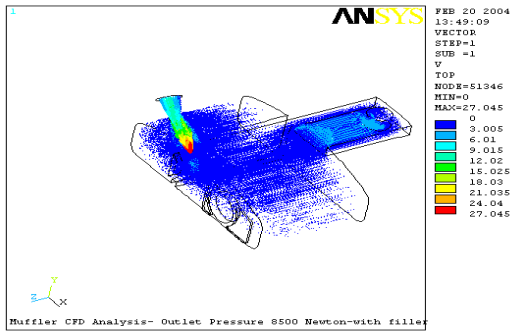
Vector plot at outlet

VECTOR PLOT FOR DESIGN 3(2)

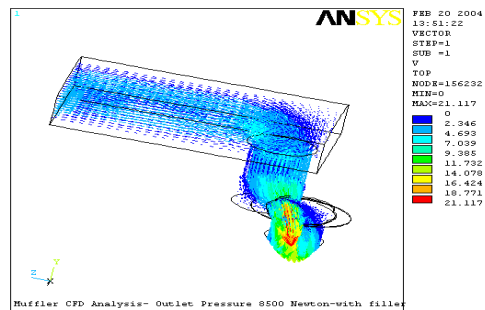


**Average Velocity at inlet
 (V_i) = 4.64 m/sec**

**Average Velocity at outlet =
 (V_o) 5.94 m/sec**



Zoomed Vector plot at inlet



Zoomed Vector plot at outlet

4. EXPERIMENTAL SET UP

Once after establishing in C F D that there is a 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

	WITH OUT SUCTION SEPARATOR	WITH SUCTION SEPARATOR
MASS FLOW	32.3 gr/min	33.2 gr/min
CAPACITY	344.15 BTU/Hr	353.72 BTU/Hr

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

MODEL		DESIGN 1	DESIGN 2	DESIGN 3(1)	DESIGN 3(2)
4.17 C.C	CAPACITY (BTU/Hr)	340	353	347	355.
	EER (BTU/Hr / W hr)	3.6	4.0	3.96	4.0
	NOISE LEVEL	<40 dBA	45 dBA	< 40 dBA	< 40 dBA
6.5 C.C	CAPACITY (BTU/Hr)	516	557	548	556
	EER (BTU/Hr / W hr)	3.65	3.97	3.94	4.0
	NOISE LEVEL	<42 dBA	47 dBA	< 42 dBA	< 42 dBA

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 there 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
2. Munjal M. C. , Acoustics of Ducts and Muffler: Wiley , New York 1987
3. John .D. Anderson , Jr 2003 ,”Computational Fluid Dynamics” Mc Graw Hill International Edition New York ,1995 .
4. International Standards ISO 3745 “Determination of Sound Power Level of Noise Source‘– Precision method for Anechoic and Semi Anechoic room.