2006

Use of CFD in Design and Development of R404A Reciprocating Compressor

Yogesh V. Birari  
*Kirloskar Copeland Limited*

Sanjay S. Gosavi  
*Kirloskar Copeland Limited*

Pavan P. Jorwekar  
*Kirloskar Copeland Limited*

Follow this and additional works at: [http://docs.lib.purdue.edu/icec](http://docs.lib.purdue.edu/icec)

[http://docs.lib.purdue.edu/icec/1727](http://docs.lib.purdue.edu/icec/1727)

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](https://engineering.purdue.edu/Herrick/Events/orderlit.html)
Use of CFD in Design and Development of R404A Reciprocating Compressor

YOGESH V. BIRARI¹*, SANJAY S. GOSAVI², PAVAN P. JORWEKAR³

¹Kirloskar Copeland Limited, Product Engineering Department, Karad, Maharashtra, India
   Phone +91 2164 241002 Fax +91 2164 241122 Email – yogeshb@kircop.com

²Kirloskar Copeland Limited, Product Engineering Department, Karad, Maharashtra, India
   Phone +91 2164 241002 Fax +91 2164 241122 Email – sanjayg@kircop.com

³Kirloskar Copeland Limited, Product Engineering Department, Karad, Maharashtra, India
   Phone +91 2164 241002 Fax +91 2164 241122 Email – pavanj@kircop.com

*Indicate Corresponding Author

ABSTRACT

For refrigeration compressor, replacing existing refrigerant by alternate refrigerant means redesigning and optimization of the various components in order to maintain / improve the compressor performance. The conventional approach involves time consuming physical experiments. This paper deals with the use of the CFD tool for reducing physical experiments and effectively the total development time of compressor, with R404A as an alternate refrigerant.

Following CFD analysis are carried out

- Thermal mapping of internal domain to predict the temperatures at various critical locations like motor winding, suction and discharge plenum, overload protector ambient.
- Pressure drops across suction and discharge passages
- Compression process simulation using dynamic meshing

Pressures, mass flow rate, velocity and flow pattern of refrigerant inside the compressor are the part of the output from simulation. The results of the simulation are validated by experiments. An approach is established to use CFD simulation, which is fast and reliable tool for design and development of reciprocating compressor with alternate refrigerants.

1. INTRODUCTION

The international and national regulations have imposed a very high demand on compressor manufacturer to design or validate the existing design schemes for alternate refrigerants which are more environmental friendly. Besides minimizing the overall environmental impact, the performance of the compressor is to be maintained or improved. The shortest way to do so is the validation of the existing design scheme.

The conventional approach of product design i.e. “Trial and Error” involves lot of time and cost for physical experimentation, also in some cases it is quite difficult to carry out such experiments. The availability of various Computer Aided Engineering (CAE) tools has now changed the whole perspective of product design process. Even though CAE alone delivers less reliable results than prototyping, if used with proper understanding of the physics of the problem; it can enable the most cost-effective assessment of many design alternatives. Expertise plays a
significant role in CAE because ability and experience is required to capture the functional behavior of components with abstract models.

Computational Fluid Dynamics (CFD) is one of the CAE tools been used in positive displacement compressor design process since from 1990 (Rodgers and Wagner, 1990). Fagotti and Possamai (2000) have demonstrated CFD as a compressor design tool. B.G. Shiva Prasad (2004) has discussed the applications, benefits and risks involved in CFD for positive displacement compressors. CFD provides enough fidelity to understand the impact of design choices, thus contributing to a deeper understanding of product behavior and problem detection before the commitment of production resources.

This paper describes the use of CFD in design and development of reciprocating compressor with R404A as an alternate refrigerant. The proposed CFD simulations are carried out for hermetically sealed reciprocating compressor. This compressor is presently used in High Back Pressure (HBP) applications like air conditioning with R22 as a refrigerant. The same compressor is proposed for Commercial Back Pressure (CBP) applications with R404A as an alternate refrigerant

2. COMPUTATIONAL FLUID DYNAMICS

The computational Fluid Dynamics (CFD) is the science of predicting the fluid flows, heat transfer and related phenomenon by solving numerically the set of governing mathematical equations i.e. conservation of mass, momentum and energy. These mathematical equations are either in form of integral or partial differential equations. CFD is the art of replacing the integral or partial derivatives in these equations with discretized algebraic form, which are further solved to obtain numbers for the flow field values at discrete points in time and/or space. There are number of discretization methods used in CFD codes. The most common and the one on which FLUENT is based, is known as the finite volume method. In this method, the domain is divided into a finite set of control volumes or cells. The equations are discretized and solved iteratively for each control volume. As a result, an approximation of the conserved variable averaged across the control volume can be obtained. This derives a full picture of the behavior of the flow.

3. MODELING GOAL – THERMAL MAPPING

The thermal energy analysis across the compressor domain is an important aspect as far as compressor performance and reliability is concerned. With change in application and refrigerant, it is vital to check the temperature distribution at various critical locations.

3.1 Computational Model and Assumptions

While selecting the computational domain, consideration was given to various issues like - What specific results are required from the CFD model and how they will be used? What degree of accuracy is required from the model? From these considerations, the domain of interest derived consisted of stator, rotor, discharge tube, suction pick-up tube, suction muffler and upper shell. Figure 1 shows the computational domain. Following assumptions are made for carrying out the analysis

- The refrigerant is considered as an incompressible fluid
- The radiation heat transfer is neglected.
- Various properties of refrigerant are considered as piecewise linear.
- A steady state condition is assumed for the simulation

3.2 Discretization

The solid modeling and geometry simplification of aforesaid components was done in view of ease of decomposition and meshing. The geometries were imported into a preprocessor where fluid and solid zones were decomposed. Fluid and solid zones were meshed with structured, block structured and unstructured scheme considering the complexity of the geometry. Other major issues which were taken into account while meshing were setup time, computational expenses and numerical diffusion. In the region of large gradient, reasonable fine grid was created in order to minimize the change in flow variable from cell to cell. Mesh quality plays significant role in the
accuracy and the stability of numerical solution. Mesh quality was checked for node point distribution, smoothness and cell shape i.e. skewness and aspect ratio. Figure 2 represents the meshed model of the domain.

3.3 Boundary Conditions

Physical properties of fluid and solids were defined using material panel. Typical properties defined are density ($\rho$), dynamic viscosity ($\mu$), specific heat ($C_p$) and thermal conductivity ($k$). Piecewise linear profiles were assumed for variation of $\rho$, $C_p$, $\mu$, and $k$ with respect to the temperature. Boundary conditions were assigned to appropriate zones. Heat sources inside the compressor like copper and iron losses from the motor, frictional losses, heat due to gas compression were considered. The copper and iron losses for the motor were obtained from the dynamometer testing and were assigned as volumetric heat sources to the respective sub-domains. Heat of compression and frictional losses were defined at the appropriate places in the domain. Velocity of suction gas and the suction gas temperature as per standard ASHRAE conditions was defined at the inlet of the domain. The outlet of the domain was defined as outflow.

3.4 Problem setup and solution

The domain needs to be solved with suitable turbulence model considering the physics encompassed in the flow. There is no single turbulence model which is universally accepted as being superior for all class of problem. The standard $k$-$\epsilon$ model was selected for its robustness, economy, and reasonable accuracy for a wide range of turbulent flows in industrial flow and heat transfer simulations (Fluent User’s Guide 2005). The turbulence parameters like turbulence intensity and hydraulic diameter were calculated and specified at appropriate zones. The solution was initialized from inlet boundary and solved for adequate number of iterations with first order scheme. Second order upwind scheme was used for rest of the iterations to get better accuracy. The convergence of the solution was judged based on scaled residuals and the surface integrals of flow variables at various locations.

4. MODELING GOAL – PRESSURE DROPS AND COMPRESSION PROCESS

The pressure drop across the suction and discharge passages has a strong influence on the performance of the compressor. Hence, the suction and discharge passages which are acoustically optimized should be optimized for the pressure drop.

4.1 Pressure drops across suction and discharge passages

The computational domain of suction passage was selected from the inlet of the pickup tube to the inlet to the cylinder bore. While for the discharge passage, it was selected from the inlet of discharge port to the exit of compressor discharge tube. The domains were meshed with block structured and unstructured mesh scheme. A known mass flow rate was imposed at the inlet and constant pressure was considered at the outlet. Turbulence was taken into account using a $k$-$\epsilon$ model. The comparison of CFD and experimental results were done.
4.2 Compression process simulation
Dynamic mesh in the FLUENT can be used for the flow where the shape of the domain is changing with time due to the motion on the domain boundaries. This capability is used to simulate the compression process with R404A as a refrigerant. The domain for this simulation was a total cylinder volume which includes the clearance volume. Top wall of the domain represents the valve plate face adjacent to cylinder bore and was considered as a rigid wall. Side wall represents the cylindrical face of the cylinder bore and was considered as a deforming wall. The bottom wall represents the piston top face and was considered as a rigid wall. The motion of bottom wall was determined by FLUENT based on the various in-cylinder input parameters like crank shaft speed, stoke, connecting rod center distance etc. Figure 3 shows the piston motion profile. The update of volume mesh is handled automatically by FLUENT at each time step based on new position of boundaries. The unsteady solution was solved with segregated solver and implicit formulation. Figure 4 shows the convergence history of volume weighted average static temperature. The results obtained were compared with the analytical temperature calculated considering the reversible adiabatic compression as given in Equation (1)

\[
\frac{T_d}{T_s} = \left( \frac{P_d}{P_s} \right)^{\frac{1}{\gamma}}
\]

\[\text{Equation (1)}\]

Figure 3: Piston Motion profile  
Figure 4: Convergence history of Static Temperature

5. RESULTS AND DISCUSSION
The solution was carried out with R404A as an alternate refrigerant and results were compared with R22 as an existing refrigerant. Table 1 shows the comparison of temperatures at various locations inside the domain. Comparison is also done for experimental and CFD result with R22. The maximum deviation between predicted and the measured temperature is 5%. These deviations are due to various assumptions in CFD models like the experimental motor losses were measured with different ambient temperature, piecewise linear refrigerant properties to name a few. The maximum increase in temperature for changed application with R404A is approximately 5% as compared to existing application with R22.

Table 2 shows the comparison of pressure drops across the suction and discharge paths. It can be seen that the overall pressure drop across the suction and discharge passage is decreased by 5% and 28% respectively with R404A as compare to that of R22. This is mainly caused by lower mass flow rate due to change in the application.

Figure 5 and Figure 6 shows the contour plot for total temperature across the domain. From the figure it can be seen that the winding temperatures in the vicinity of suction tube are lower than the average winding temperature where
as those are higher on bottom winding. Figure 7 and Figure 8 shows contours of total temperature along with path lines for the whole domain. Figure 9 and Figure 10 show the contours of total pressure across the discharge path with R22 and R404A respectively.

Table 1 - Comparison of Experimental and CFD thermal mapping results

<table>
<thead>
<tr>
<th>S/N</th>
<th>Location</th>
<th>Temperature with R22 (K)</th>
<th>Temperature with R404A (K)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>Experimental</td>
<td>CFD</td>
</tr>
<tr>
<td>1</td>
<td>Top Winding</td>
<td>Location1</td>
<td>352</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Location2</td>
<td>363</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Location3</td>
<td>353</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Location4</td>
<td>345</td>
</tr>
<tr>
<td>2</td>
<td>Bottom Winding</td>
<td>Location5</td>
<td>390</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Location6</td>
<td>378</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Location7</td>
<td>370</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Location8</td>
<td>372</td>
</tr>
<tr>
<td>3</td>
<td>Suction Plenum</td>
<td></td>
<td>345</td>
</tr>
<tr>
<td>4</td>
<td>Upper Shell</td>
<td></td>
<td>339</td>
</tr>
</tbody>
</table>

Table 2 - Comparison of Pressure Drop

<table>
<thead>
<tr>
<th>S/N</th>
<th>Location</th>
<th>R22 (Pa)</th>
<th>R404A (Pa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Suction Path</td>
<td>2600</td>
<td>2478</td>
</tr>
<tr>
<td>2</td>
<td>Discharge Path</td>
<td>54280</td>
<td>38977</td>
</tr>
</tbody>
</table>

Figure 5: Total Temperature across domain with R22
Figure 6: Total Temperature across domain with R404A
Figure 7: Total Temperature and path lines with R22

Figure 8: Total Temperature and path lines with R404A

Figure 9: Total Temperature on upper shell with R22

Figure 10: Total Temperature on upper shell with R404A

Figure 11: Total Pressure in discharge path with R22

Figure 12: Total Pressure in discharge path with R22
6. CONCLUSIONS

The existing design scheme with alternate refrigerant was investigated by using the commercial CFD package – FLUENT. The scheme with alternate refrigerant was checked for the heat balance across the compressor domain and suction and discharge path pressure drops. The simulation was performed with R22 as an existing refrigerant and R404A as alternate refrigerant at ASHRAE rating conditions. The CFD and experimental results are in good agreement. The design changes in the existing compressor were worked out based on the CFD simulation results, resulting in reduced number of prototypes for achieving intended results.

NOMENCLATURE

\[ \begin{align*}
\rho & \quad \text{Density (kg/m}^3) \\
\mu & \quad \text{Dynamics viscosity (Pa-s)} \\
C_p & \quad \text{Specific heat (J/kg-K)} \\
k & \quad \text{Thermal conductivity (W/m-K)} \\
\varepsilon & \quad \text{Turbulence dissipation rate (m}^2\text{/s}^3) \\
T & \quad \text{Temperature (K)} \\
P & \quad \text{Pressure (Pa)} \\
\gamma & \quad \text{Ratio of specific heats, } C_p/C_v \text{ (Dimensionless)} \\
k & \quad \text{Turbulence kinetic energy (m}^2\text{/s}^2) 
\end{align*} \]

REFERENCES


Fluent 6.2, 2005 User’s Guide,

ACKNOWLEDGEMENT

The authors gratefully acknowledge the continual support provided by Product Engineering Dept (PED) and Technology Support Dept (TSD) staff.