2002

Thermal Mapping Of Hermetically Sealed Compressors Using Computational Fluid Dynamics Technique

R. C. Chikurde
Kirloskar Copeland Ltd.

E. Longanathan
Kirloskar Copeland Ltd.

D. P. Dandekar
Kirloskar Copeland Ltd.

S. Manivasagam
Kirloskar Copeland Ltd.

THERMAL MAPPING OF HERMETICALLY SEALED COMPRESSORS USING COMPUTATIONAL FLUID DYNAMICS TECHNIQUE

R. C. Chikurde, E. Loganathan, D. P. Dandekar, S. Manivasagam*
Product Engineering Department and CAE Division,
Kirloskar Copeland Ltd.,
Karad – 415 110, India.
Email – manivasagam@kircop.com

ABSTRACT

The complex fluid flow and heat transfer phenomena in hermetic compressors is very difficult to analyze theoretically because of the assumptions made while solving the problem and insufficient understanding of the physics involved. There is a need to thoroughly understand such phenomena to target the efficiency improvements of the compressor to meet current high EER requirements. This paper deals with the use of Computational Fluid Dynamics (CFD), a highly sophisticated numerical technique based on Finite Control Volume formulation, to analyze the entire compressor domain.

NOMENCLATURE

ρ	Density
∇	Vector operator
V	Vector velocity
t	time
u	Velocity component in x-direction
v	Velocity component in y-direction
w	Velocity component in z-direction
p	Pressure
p_{op}	Operating pressure
τ	Shear stress
f	Body force per unit mass acting on fluid element
e	Internal energy per unit mass
q	Rate of volumetric heat generation per unit mass
k	Thermal conductivity
μ	Viscosity
C_p	Specific heat
T	Temperature
n	Polytropic exponent
R	Universal gas constant
M_w	Molecular weight of the gas
k	Turbulent kinetic energy
ε	Turbulence dissipation rate

INTRODUCTION

Many researchers have addressed thermal energy analysis of reciprocating compressors using numerical modeling to predict the performance in terms of power input, mass flow rate, heat transfer rates and temperatures of internal compressor components [1].

This paper attempts to discuss more about the application of the results obtained by simulating a whole compressor model using commercially available CFD code, FLUENT. For this analysis, whole compressor model is selected as the computational domain and meshed in GAMBIT, preprocessor for FLUENT. The material and boundary conditions are applied in FLUENT solver and the domain is solved to satisfy continuity, momentum and energy conservation equations. This powerful tool along with faster and robust digital computers makes it possible to predict velocity, pressure and temperature distribution in any plane across the whole domain and also to visualize the overall flow pattern, separation etc. along the gas flow paths [2].

The results of the numerical simulation are validated by using an experimental set-up for thermal mapping of the compressor with conventional thermocouples. This comparison indicates that the numerical solution agrees with the experimental results with sufficient accuracy. Thus, use of CFD as a design and research tool in analyzing complex physical situations found in compressors provides a means to interpret the experimental conditions.

COMPUTATIONAL FLUID DYNAMICS – BASIC GOVERNING EQUATIONS

CFD is based on three fundamental physical principles viz. conservation of mass, Newton's second law of motion and conservation of energy. By applying these principles to a fluid element or control volume, one gets continuity, momentum and energy equations which govern the flow of fluid. These equations are applicable for an unsteady, three dimensional, compressible and viscous flows [3].

Continuity equation:

$$\frac{\partial \rho}{\partial t} + \rho \nabla . V = 0 \qquad \text{.....(1)}$$

Momentum equations:

$$\frac{\partial(\rho u)}{\partial t} + \nabla.(\rho u V) = -\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} + \rho f_x \qquad \text{.....(2)}$$

$$\frac{\partial(\rho v)}{\partial t} + \nabla.(\rho v V) = -\frac{\partial p}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} + \rho f_y \qquad \text{.....(3)}$$

$$\frac{\partial(\rho w)}{\partial t} + \nabla.(\rho w V) = -\frac{\partial p}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} + \rho f_z \qquad \text{.....(4)}$$

For a viscous flow, the above x, y and z components of the momentum equations are also called the ‘ Navier – Stokes equations ’.

Energy equation:

$$\frac{\partial}{\partial t}\left[\rho\left(e+\frac{V^2}{2}\right)\right]+\nabla.\left[\rho\left(e+\frac{V^2}{2}\right)V\right]=\rho\dot{q}+\frac{\partial}{\partial x}\left(k\frac{\partial T}{\partial x}\right)+\frac{\partial}{\partial y}\left(k\frac{\partial T}{\partial y}\right)+\frac{\partial}{\partial z}\left(k\frac{\partial T}{\partial z}\right)$$

$$-\frac{\partial(up)}{\partial x}-\frac{\partial(vp)}{\partial y}-\frac{\partial(wp)}{\partial z}+\frac{\partial(u\tau_{xx})}{\partial x}+\frac{\partial(u\tau_{yx})}{\partial y}+\frac{\partial(u\tau_{zx})}{\partial z}$$

$$+\frac{\partial(v\tau_{xy})}{\partial x}+\frac{\partial(v\tau_{yy})}{\partial y}+\frac{\partial(v\tau_{zy})}{\partial z}+\frac{\partial(w\tau_{xz})}{\partial x}+\frac{\partial(w\tau_{yz})}{\partial y}+\frac{\partial(w\tau_{zz})}{\partial z}+\rho f.V \quad \text{.....(5)}$$

The above equations form a coupled system of non-linear partial differential equations, which are difficult to solve analytically. To simplify the computational efforts, the small scale turbulent fluctuations in Navier – Stokes equations are modeled using Reynolds averaging or filtering approach. The resulting set of governing equations are integrated on each control volume to construct algebraic equations for discrete dependent variables such as velocities, pressure, temperature etc. Additional transport equations are also solved when the flow is turbulent.

SELECTION OF THE DOMAIN AND ASSUMPTIONS

The proposed numerical simulation is carried out for a hermetically sealed reciprocating compressor used for 1.5 Ton Air-conditioning applications. The geometry of the entire compressor is modeled using I-DEAS Master Series. Because of the geometrical complexity of the domain, the model is simplified by removing the unnecessary fillets and restrictions so that it is easy to mesh the domain in preprocessing stage. The domain of interest consists of compressor shell, stator stack along with copper winding, rotor, air gap between stator and rotor, crank case, crankshaft, mufflers, valve plate with suction and discharge ports, cylinder head including suction and discharge plenum cavities and discharge shock loop. The suction and discharge tubes are also modeled in order to allow the refrigerant gas to enter and exit the domain.

The suction gas enters the compressor shell through suction tube and the gas stream is directed towards the bottom winding and lower portion of stator stack. The gas then enters the pick-up tube through the rectangular aperture provided near the top of upper winding, thus serving the purpose of cooling the upper winding also. The pick-up tube leads the suction gas into the crankcase cavity and then it enters the suction plenum. The discharged gas from the discharge plenum in the cylinder head comes out of the compressor through discharge muffler and shock loop. The incorporation of both, suction and discharge gas accounts for the heat transfer from the discharge gas at a high temperature to the suction gas through cylinder head walls. The coupling between both the gas paths is taken into account by the polytropic compression law as stated in following assumptions.

Following assumptions are made for carrying out the analysis –

1) The refrigerant gas is assumed to be ideal gas, and follows the polytropic change. The discharge gas temperature is calculated by using isentropic relation as below [4] -

$$\frac{Td}{Ts} = \left(\frac{p_d}{p_s} \right)^{\frac{(n-1)}{n}}$$

where, subscripts 's' and 'd' corresponds to the suction and discharge gas conditions.

2) The analysis is carried out assuming convection and conduction heat transfer. The radiation heat transfer is neglected.
3) The heat losses from the copper coil, stator lamination and rotor are assumed to be volumetric heat sources. (Watts/cu. m)
4) The walls are smooth everywhere.

DISCRETIZATION OF THE DOMAIN

The entire compressor geometry is translated into GAMBIT, which is used as a preprocessor for FLUENT solver. It is necessary at this stage, to decompose the geometry into solid and fluid zones. This is a tedious task and one has to be careful while solid modeling the compressor parts. The decomposition process requires tight tolerances in the topology of the computational domain. The domain is discretized with hexahedral, tetrahedral, wedge and pyramid elements. It is very difficult to create structured mesh in the entire domain and one has to use other meshing schemes such as unstructured or hybrid mesh depending on the domain complexity. The quality of the elements is checked in terms of equi-angle skew and the aspect ratio. The equi-angle skew indicates the deviation of an element shape from its true geometrical shape. e.g. equi-angle skew is zero for a tetrahedral element having 60^o angle between all of its side faces.

While meshing, additional care has to be taken so that key flow features such as high velocity gradient, pressure gradient and turbulence quantities like turbulent kinetic energy etc. would properly be resolved in order to get sufficiently accurate results. The domain inlet, exit and other surface boundary conditions along with solid and fluid zones are defined. The total number of elements in the computational domain are nearly 1.8 Million. The mesh is then imported into the FLUENT solver, for assigning material properties and boundary conditions.

MATERIAL PROPERTIES AND BOUNDARY CONDITIONS

The proper material and its properties are defined for fluid as well as solid zones such as density, viscosity, specific heat, thermal conductivity etc. which are essential for solving basic governing equations. The gas density is calculated using the ideal gas law for incompressible flow as follows [5] -

$$\rho = \frac{p_{op}}{\left(\frac{R}{M_w}\right)T}$$

Piecewise linear profiles are assumed for the variation of μ, Cp and k with respect to the temperature. While assigning boundary conditions, some inputs are needed such as mass flow or flow velocity at the inlet, external heat transfer coefficient, various heat sources such as frictional losses, copper and iron losses in motor, heat due to gas compression etc. The values of these parameters can either be calculated theoretically or obtained experimentally. For the present simulation, the copper and iron losses are obtained from the dynamometer testing and the parameters like external heat transfer coefficient are calculated using standard correlations [6]. The heat sources are defined as volumetric sources to the respective sub-domains. The calculated mass flow rate and suction gas temperature are defined at the inlet. The outflow boundary condition is used at the outlet, which assumes that all the gradients except pressure are zero and the flow is in a fully developed condition. The operating pressure is set according to standard ASHRAE conditions.

PROBLEM SET-UP AND SOLUTION

There are two numerical methods for solving the governing equations – segregated and coupled solver formulation. Segregated solver solves the continuity, momentum and energy equations sequentially whereas, coupled solver solves the equations simultaneously. Implicit formulation is used for linearization of equations. Choosing the appropriate turbulence model is an important step, which depends on the nature of gas flow inside the suction and discharge paths, i.e. highly turbulent flow, rotating or flow with strong coupling between the turbulent quantities and the mean flow quantities. The standard k-ε model is used in the present simulation which is robust, economic and gives reasonable accuracy for a wide range of turbulent flows [5]. The turbulence parameters such as turbulence intensity and hydraulic diameter are specified. The second order upwind scheme is used for the discretization of the convection terms and pressure interpolation. The domain is initialized based on inlet boundary and the simulation is carried out in steady state condition for sufficient number of iterations to get the convergence. The predicted temperatures and pressure drop values are shown in Table 1.

EXPERIMENTAL SET-UP

The results of the numerical simulation are validated by using an experimental set-up for temperature mapping of the compressor with conventional thermocouples. One end of the thermocouple wires is placed at the required point inside the compressor and the other end is connected to multi-selector switch. The compressor is mounted on calorimeter test rig and the test is carried out under standard test conditions as per ASHRAE. The values of the temperatures at various points are noted during normal steady state running of the compressor. The comparison between experimental and numerical results is shown in Table 1.

RESULTS AND DISCUSSION

It is well known that the most important parameters affecting the efficiency of the compressor are the suction gas superheating and the flow and energy losses occurring inside the suction and discharge gas paths. The predicted pressure drop, flow velocities and temperatures at motor winding, stator stack, refrigerant gas in suction and discharge plenums, discharge line temperature etc. help in understanding how the flow physics changes inside the compressor. The numerical model clearly shows the temperature and pressure gradients along the suction and discharge gas paths. CFD also predicts the flow separation, back flow and eddies at abrupt cross sectional changes in gas paths. This helps in assessing the performance of existing as well as new compressor models for higher efficiency, optimizing gas paths for minimum flow losses and designing efficient motor with better cooling.

Fig. 1 shows velocity vector plot in the entire domain of the compressor. It can be seen that the suction gas stream comes in contact with the lower winding of the motor providing better cooling in lower portion of the motor. Fig. 2 shows Path line plot for visualizing the particle flow path in the computational domain.

Fig. 3 shows temperature contour plot across a vertical plane through entire domain and Fig. 4 shows temperature distribution on stator stack and winding, facing the suction gas. Fig. 5 and 6 show the temperature variation on motor surface from bottom winding to top winding. As can be seen from Fig. 5, CFD predicts lower stack temperature than bottom winding as the suction gas stream directly impinges on the lower portion of the stator stack. The maximum temperature occurs at top winding. On the opposite side of suction gas inlet, the bottom winding temperature and stack temperature are on higher side as shown in Fig. 6. The maximum temperature occurs at approximately half of the stator stack.

CONCLUSIONS

This paper demonstrates the strength of Computational Fluid Dynamics in simulating flow and thermal characteristics of the overall compressor in steady-state conditions. A numerical model for 1.5 Ton A/C compressor is developed and analyzed using commercial CFD package, FLUENT. The entire computational domain is solved for conjugate (i.e. conduction and convection) heat transfer and the results are seen across different planes. It is seen from the table that the numerical results are in good agreement with the experimental results and the assumptions made to solve the domain are reasonable.

The simulation procedure can also be extended for predicting the motor winding temperature for different field conditions such as loss of refrigerant charge, blocked fan condition, maximum load, start and run winding short etc. This data could be used in selecting internal overload protector (OLP), which protects the motor from failing under adverse conditions. This reduces iterative testing for OLP selection and the need for building prototypes for this purpose.

Besides, CFD is used here at the design stage of the compressor, thereby reducing the number of prototypes for trial and error and hence the reduction of total design cycle time. The visualization tools like velocity vector plots, temperature and pressure contours are used to streamline the gas flow by modifying and improving flow passages in plenums, mufflers and shock loop. The developed numerical model can, further, be used to predict the effect of design changes and load changes over gas superheating, temperature distribution and pressure losses as well as compressor efficiency.

ACKNOWLEDGEMENTS

The authors take this opportunity to express their sincere thanks to the KCL management for their continuous encouragement as well as to the members of PED and CAE Division for their active support. The authors also express their sincere thanks to the Technology Support Department for their experimental support. The authors are also very thankful to Fluent India Pvt. Ltd., Pune for their excellent support during various stages of the project.

REFERENCES

1. M. L. Todescat, F. Fagotti, A. T. Prata and R. T. S. Ferreira, 1992, " Thermal Energy Analysis in Reciprocating Hermetic Compressors ", Int. Comp. Eng. Conf. at Purdue, 1419 – 1427
2. F. Fagotti & F. C. Possamai, 2000, " Using Computational Fluid Dynamics as a Compressor Design Tool ", Int. Comp. Eng. Conf. at Purdue, 137 - 144
3. John D. Anderson, Jr., " Computational Fluid Dynamics – The Basics with Applications ", McGraw-Hill International Editions, New York, 1995
4. Suefuji, K. and Nakayama S., 1980 " Practical Method for Analysis and Estimation of Reciprocating Hermetic Compressor performance ", Int. Comp. Eng. Conf. at Purdue, 15 - 23
5. Fluent User's Guides, Volumes 1 to 4, Fluent Inc., Lebanon, NH, 1998
6. Incropera, F. P. and DeWitt, D. P., " Fundamentals of Heat and Mass Transfer ", Second edition, John Wiley and Sons, New York, 1985

Table 1:- Comparison between experimental and numerical results for 1.5 Ton A/C compressor for ASHRAE standard test conditions

S/N	Measurement location for temperature in K	Average experimental results	Average Numerical results from FLUENT
1	Top winding	352	356.5
2	Bottom winding	347	346
3	Stator stack	Not measured	364
4	Rotor stack	Not measured	360
5	Compressor ambient (Shell gas temp)	345	341
6	Gas temp at suction pick-up tube inlet	348	342
7	Top shell	335.5	331.5
8	Bottom shell	340	341.5
9	Suction plenum gas	354	356
10	Discharge plenum gas	408	405
11	Discharge line	388	395
12	Oil sump	346	349
	Pressure drop values in Pascal		
1	Pressure drop in suction gas path	Not measured	8871.3
2	Pressure drop in discharge gas path	Not measured	48275.8

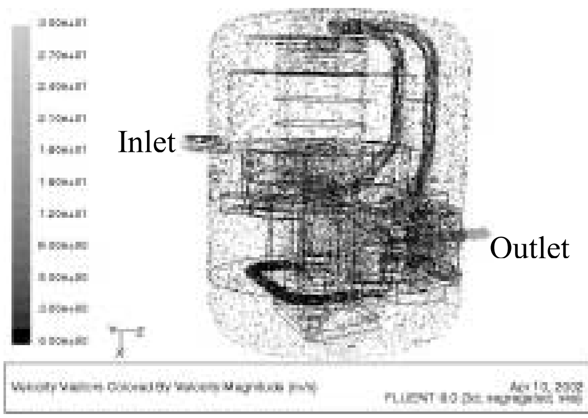

Fig. 1 Velocity vector plot in entire compressor domain

Fig. 2 Path line plot released from the inlet

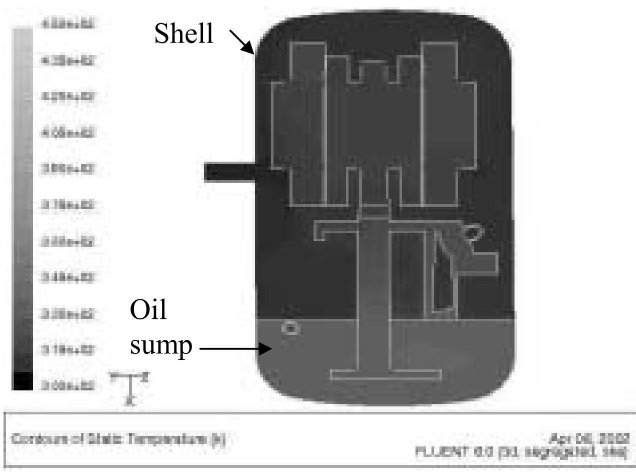

Fig. 3 Temperature contour plot across a vertical plane through entire domain

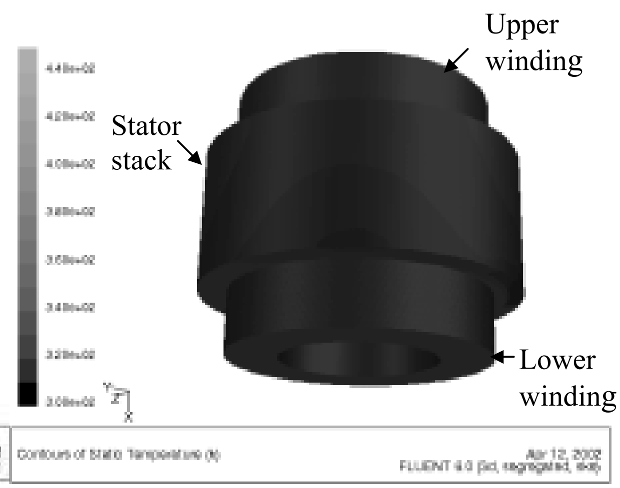

Fig. 4 Temperature distribution on stator stack and windings

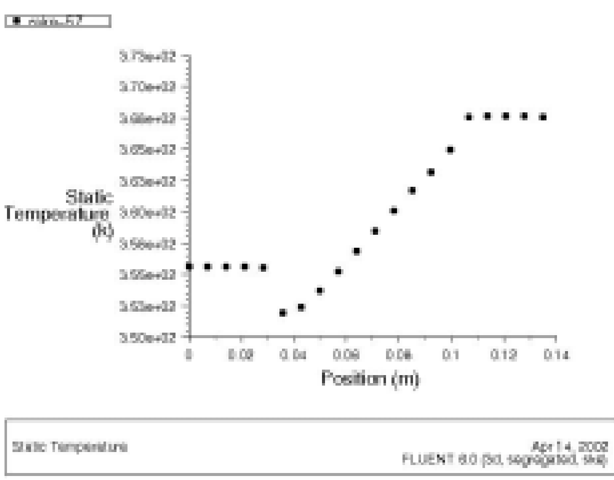

Fig. 5 Temperature variation along the motor surface facing the suction gas inlet

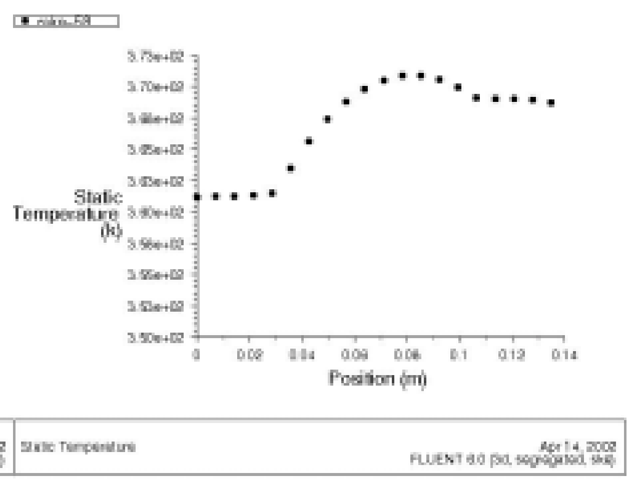

Fig. 6 Temperature variation along the motor surface opposite to suction gas inlet