

# CFD ANALYSIS OF CROSS FLOW AIR TO AIR TUBE TYPE HEAT EXCHANGER

Vikas Kumar<sup>1\*</sup>, D. Gangacharyulu<sup>2\*</sup>, Parlapalli MS Rao<sup>3</sup> and R. S. Barve<sup>4</sup>

<sup>1</sup>Centre for Development of Advanced Computing, Pune University Campus, Pune, India

<sup>2</sup>Thapar Institute of Engineering & Technology, Patiala, India

<sup>3</sup>Nanyang Technological University, Singapore

<sup>4</sup>Crompton Greaves Ltd, Kanjur Marg, Mumbai, India

Email: [vikaskumar\\_gupta@yahoo.com](mailto:vikaskumar_gupta@yahoo.com), [gangacharyulu@rediffmail.com](mailto:gangacharyulu@rediffmail.com)

Date: June 2003

Computers & Operating System: Pentium III PC with Windows 98,  
Parallel PHOENICS 3.4 on Sun Solaris

PHOENICS version: 3.3 & 3.4

## Abstract

A three dimensional numerical simulation study has been carried out to predict air flow and temperature distribution in the tube type heat exchanger associated with large electrical motor. Due to symmetry in geometrical construction, a section of heat exchanger has been considered for CFD analysis by using PHOENICS software. The  $k - \epsilon$  turbulence model has been used to solve the transport equations for turbulent flow energy and the dissipation rate. The outlet temperature of cold & hot air predicted by the CFD simulation has close agreement with experimental results. Further, the performance of parallel PHOENICS on PARAM 10000 is reported in the present work.

Keywords: CFD analysis, CACA, Large motor, Tube type heat exchanger, Simulation, Parallel PHOENICS

## Contents

1. Introduction
  2. Description of the Phenomenon Simulated
  3. CFD Model
  4. Results & Discussions
  5. Parallel PHOENICS Computation
  6. Conclusions
  7. Acknowledgements
- References  
Nomenclature

---

\* Corresponding authors

## **1.0 Introduction**

The closed air circuit air cooled (CACCA) electrical motors are used in various industries for higher rating (500 kW and above) applications. The equipment generates heat due to the energy losses in the windings at various electrical loads under operating condition. The cold air is circulated in the motor to remove the heat generated in the motor. The hot air generated in the motor is cooled by using air to air tube type cross flow heat exchanger because of its added advantages, e.g. low maintenance as well as operational costs over water cooled systems [1 & 2]. The performance of CACA motor is influenced by the rise of windings temperature, therefore, the temperature rise of the equipment is restricted for efficient working and long life of the system. The motor designers are interested to know the temperature distribution of air in the heat exchanger and pressure drop across the tube bundle at various operating parameters, e.g., different hot & cold air temperatures and fluid (hot & cold) flow rates. The objective of the present work is to study the pressure, air flow & temperature distribution in the heat exchanger through numerical simulation studies.

## **2.0 Description of the Phenomenon Simulated**

The heat exchanger consists of a tube bundle housed in a rectangular enclosure. The geometry of the heat exchanger is shown in Figure 1. The cold air flows inside the tubes and the external surfaces of tubes are exposed to hot air coming from the motor. The exchanger is divided into four sections, which are separated by internal baffles and partition plate. The baffle is used to guide the flow path of hot air. The end separating plate separates cold air from hot air, whereas the partition plate divides heat exchanger into two sections and distributes hot air in the both sections of heat exchanger. The hot air leaving from the motor enters into the middle two sections (second & third) of heat exchanger and exchanges its thermal energy with the tube bundle as well as cold fluid inside the tube. The cooled hot air leaving from the first and the fourth sections of heat exchanger is recirculated into the motor. The cold air exiting from the tube end is discharged to the atmosphere.

The generalized form of transport equation governing fluid flow & heat transfer can be expressed as [3]:

$$\frac{\partial (\rho\phi)}{\partial t} + \frac{\partial}{\partial x_j} \left[ \rho U\phi - \Gamma_\phi \frac{\partial \phi}{\partial x_j} \right] = S_\phi \quad (1)$$

The continuity, momentum and energy equations are solved using finite volume based PHOENICS CFD software under the steady state and turbulent flow of air with an ideal gas equation of state.

### 3.0 CFD Model

To simplify the problem domain, the motor has not been included in the CFD model. The geometrical details of heat exchanger considered for CFD analysis are given in Table 1.

Table 1: Geometrical details and boundary conditions of heat exchanger

Sl. No.	Description	Unit	Value/Type
1.	Overall dimension	mm	1760 x 100 x 765
2.	Dimension of cold air inlet	mm	100 x 765
3.	Number of hot air inlet	-	2
4.	Dimension of hot air inlet	mm	398 x 100
5.	Tube inner diameter	mm	22
6.	Tube outer diameter	mm	26
7.	Tube length	mm	1610
8.	No. of tubes	-	27
9.	Transverse pitch	mm	61
10.	Longitudinal pitch	mm	41
11.	Number of baffles	-	2
12.	Dimension of baffle	mm	5 x 100 x 660
13.	Number of end separating plates	-	2
14.	Dimension of end separating plates	mm	10 x 100 x 765
15.	Number of partition plate	-	1
16.	Dimension of partition plate	mm	5 x 100 x 765
17.	Material of baffle, separating and partition plate	-	Mild steel
18.	Material of tube	-	Electric resistance welded (ERW)

### 3.1 Modeling Considerations

1. The geometry has symmetry in breadth wise, therefore a section of heat exchanger consisting of 9 rows & 3 columns has been considered for analysis. Each column has 9 tubes.
2. The tube is modeled as solid blockage, whereas, the inner volume of tube has been modeled as blockage with gaseous properties to allow the ambient air to pass through it.
3. Conduction takes place from the tube wall & convection takes from the surface of tube. The partition plate and baffle participate in heat transfer. Temperature & flow distributions have been considered to be three dimensional in nature.
4. The Reynolds number of cold fluid at inlet is 9,414. The standard, two equation k- $\epsilon$  turbulence model has been considered, which solves two additional transport equations for the turbulent kinetic energy and the dissipation rate with the corresponding turbulence production and dissipation sources [4].
5. The hybrid differencing scheme has been used.

### 3.2 Assumptions in modeling

1. Heat is lost from the top surface of heat exchanger by natural convection. The heat transfer coefficient at the top surface has been assumed as  $5 \text{ W/m}^2\text{-K}$ .
2. Heat loss by radiation from heat exchanger surface is neglected.

### 3.3 Boundary Conditions

The inlet velocity & temperature are set to a constant value. The turbulence intensity at the inlet is set as 2%. The relative pressure value is set to zero at the outlet cell surface. The operating boundary conditions of heat exchanger are presented in Table 2

Table 2: Operating boundary conditions of heat exchanger

Sl. No	Input parameters	Unit	Value
1.	Temperature of cold air	$^{\circ}\text{C}$	35
2.	Temperature of hot air	$^{\circ}\text{C}$	63
3.	Volumetric flow rate of cold air	cuft/minute (cum/minute)	388 (10.98)
4.	Volumetric flow rate of hot air	cuft/minute (cum/minute)	228.80(6.48)

### **3.4 Grid Generation**

The mesh has been generated in the Cartesian co-ordinate system. The grid independence test has been conducted and the grid has been refined in tube wall region. The grid resolution has kept as low as 1 mm in the y & z direction to treat the small wall thickness of the tube as solid blockage. The generated grids are shown in Fig. 2. The distribution of cells in the three directions are given below:

X Direction	:	55
Y Direction	:	48
Z Direction	:	232

The total number of cells in computational domain is 6,12,480.

### **3.5 Relaxation**

The following relaxation factors for the various variable have been considered:

P1 - pressure, linear relaxation (LINRLX), 1

U1 - velocity vector in X-axis, false-time-step relaxation (FALSDT), 0.1

V1 - velocity vector in Y-axis, false-time-step relaxation (FALSDT), 0.1

W1- velocity vector in Z-axis, false-time-step relaxation (FALSDT), 0.1

TEM1 - Temperature, false-time-step relaxation (FALSDT), 1.0

The relaxation parameters of other variables have been kept as the default values.

### **3.6 Convergence**

The convergence criteria considered in the present investigation included stabilization of variables at different monitoring points as well as the mass & energy balance. The number of sweeps needed to get the convergence was 1000. The difference in the incoming & outgoing mass flux was found to be 0.009 % of the reference mass inflow. The difference in the incoming & outgoing energy flux was found to be 0.00210 % of the reference energy inflow.

## **4.0 Results & Discussions**

The observations of CFD simulation results are discussed below for the boundary conditions as mentioned in Table 2.

### **4.1 Pressure distribution in heat exchanger**

The pressure contour of heat exchanger in a vertical plane along its length in X-Z plane is shown in Fig. 3. The high pressure zone has been observed nearby the separating plate where cold fluid is separated from the hot fluid. The highest pressure region has been observed nearby the top of separating plate, which may be due to large change in the momentum of cold fluid caused by the plate. The pressure of cold fluid in the tube is higher at inlet and decreases as it moves towards outlet. The high pressure gradient along the tube length is due to friction between air and the tube wall. A small amount of negative pressure is observed nearby the outlet of cold fluid due to jet effect of cold fluid exiting from the tube.

The pressure gradient is gradual in the hot stream. The pressure drop across tube bundle is about (35.7 Pa) as compared to the pressure drop in tube side along its length, which may be of the order of 202 Pa. At the inlet of hot air, more pressure is observed due to tube bundle and the pressure decreases, as the hot air moves up. In the second section, the pressure is low near the exit from 2<sup>nd</sup> section to first section. The pressure decreases from top to bottom in the first section. The similar behaviour is observed in the 3<sup>rd</sup> & 4<sup>th</sup> section of the heat exchanger

### **4.2 Velocity distribution in heat exchanger**

Fig. 4 shows fluid flow pattern in a vertical plane along its length in X-Z plane. At the inlet of cold fluid, the velocity of cold fluid varies in all the three directions because of variation in pressure caused by the wall resistance of end separating plate. The inlet stream hits the separating plate at high velocity, most of the air passes through the tubes, and a part of fluid moves downward along the plate and flows back towards entrance area. In the X-direction, the velocity of cold fluid increases nearby the entry to the tube

due to sudden contraction of large inlet size into small tube diameter size. The velocity of air in the tube decreases gradually from the tube entry to the exit of tube. The dead zone (lowest velocity) has been observed at the top corner of 1<sup>st</sup> & 4<sup>th</sup> section. Inside the heat exchanger, the velocity increases nearby the exit from 2<sup>nd</sup> section to 1<sup>st</sup> section and 3<sup>rd</sup> section to 4<sup>th</sup> section due to throat effect.

### **4.3 Temperature distribution in heat exchanger**

Fig. 5 shows the temperature distribution of heat exchanger in a vertical plane along its length in X-Z plane. The temperature plot reveals that the hot air loses heat as it moves upward and the heat is gained by the cold fluid. Inside the tube, the temperature of cold air increases along its length as it picks up the heat from hot fluid while moving towards outlet. At the outlet, the exit air coming from bottom tubes are hotter than that of the exit air coming from top tubes, as the heat sources are located at the bottom. The hot air entrapped in the tube bundle is cooled by the cold air flowing inside the tubes. Some of the hot air, which moves upward in the side passage away from the tube bundle, does not get cooled. The side hot air mixes with the cooled hot air coming from the top of tube bundle, because of which the top surface of 2<sup>nd</sup> and 3<sup>rd</sup> section shows higher temperature as compared to the middle portion. The temperature distribution of tube bundle across the heat exchanger in the middle of second section in Y-Z plane is shown in Fig. 6. The temperature drop of hot air in the 1<sup>st</sup> section of heat exchanger is high (18.3 °C) because of high temperature difference between cold air and hot air. In the 4<sup>th</sup> section, the temperature drop of hot air is low (13.45 °C). The reason behind this observation is that the cold air has already gained heat from 2<sup>nd</sup> section, therefore its capacity to extract further heat from hot air in 3<sup>rd</sup> section has been reduced. This non uniform temperature distribution pattern of air limits the performance of heat exchanger and in turn the performance of electrical motor. The similar pattern has been obtained in the experiments. The results obtained from CFD simulations have been compared with the experimentally measured values obtained from heat run test, which is shown in Fig. 7. The cold & hot air outlet temperature has been measured at the centre of different outlets. The hot air temperature at the inlet of heat exchanger has been computed based on the energy balance criterion.

To know the effect of hot air inlet temperature on temperature distribution in the heat exchanger, a simulation study has been carried out by varying the inlet temperature of 2<sup>nd</sup> section from 63°C to 61 °C and that of the third section from 63 °C to 65 °C, while keeping the other boundary and geometrical conditions as the same. It has been observed that there is marginal variation in pressure and velocity distribution, however temperature distribution changes, which is shown in Fig. 8. A comparison of outlet temperature is made at the middle plane (Y axis) for the above cases with respect to experimental values and its numerical values are tabulated in Table 3.

Table 3: Comparison of air temperature prediction at various outlets

Sl. No	Inlet temperature, °C			Outlet temperature, °C			Remarks
	Cold air	Hot air 2 <sup>nd</sup> section	Hot air 3 <sup>rd</sup> section	Hot air 1 <sup>st</sup> section	Hot air 4 <sup>th</sup> section	Cold air	
1.	34.4	63	63	41.9	51.8	46.8	Experimental
2.	34.4	63	63	44.70	49.55	43.68	CFD Simulation
3.	34.4	61	65	43.68	50.9	44.32	CFD Simulation

It has been observed that the results of second simulation have closer value to experiments, which verifies the present analysis and represents the situation in realistic way. In actual situation the first half of motor gets cooled properly whereas the cooling of motor in the second half is not satisfactory and it has major concern for the designer. The future work will focus on the effect of baffle & partition plate for better thermal performance of heat exchanger.

## 5.0 Parallel PHOENICS Computation

MPI (Message pass interface) based Parallel PHOENICS 3.4 has been ported on PARAM 10000 supercomputer developed at C-DAC, Pune. It has 40 compute nodes. Each node has 4 Ultra SPARC-450 CPUs @ 300 MHz with 2MB external cache. The operating system is Sun Solaris 2.6. The heat exchanger problem has been run on various processors using parallel PHOENICS. The scalability of the code is shown in Fig. 9. The



computational time decreases as the number of processor increases. The computational result shows that the parallel computing will have greater application in the CFD simulation for various industrial problems.

## **6.0 Conclusions**

A method for predicting the pressure, velocity & temperature distribution in the tube type heat exchanger associated with CACA large motor has been developed using PHOENICS CFD software. The result predicted by simulation study has been compared with the experimental values. A good consensus is observed. The outlet temperature of hot & cold air has been found to be close to the experimental values. The simulated results predict the temperature distribution reasonably at different locations of heat exchanger. The CFD model may be used to optimize its thermal performance by varying the location of baffles & partition plate in the heat exchanger and in turn improve the performance of electrical motors. The parallel PHOENICS can be used to reduce the design cycle of equipment due to fast computation.

## **7.0 Acknowledgements**

The authors gratefully acknowledge the support provided by M/s Thapar Centre for Industrial Research & Development, Patiala, India for providing the necessary facilities to carry out this project and M/s Crompton Greaves, Mumbai, India for providing the funds in addition to drawing, design data and experimental results for this project. The authors also thanks to the support provided by M/s CHAM, U.K. for technical help. Finally, the authors wish to thank the management of Centre for Development of Advanced Computing (C-DAC), Pune for providing the facility to use PARAM 10000 for running parallel PHOENICS and extending encouragement for writing this paper.

## **References**

1. Paikert, P., "Air cooled heat exchanger: thermal and hydraulic design of heat exchanger", Vol. 3, Hemisphere Publishing Corporation, 1986.
2. Saunders, E.A.D., "Heat exchangers: selection, design, and construction", Longman Scientific & Technical, New York, 1988.

3. Spalding D.B., "The mathematical basis of PHOENICS", PHOENICS 3.5 manual, CHAM, UK , [http://www.cham.co.uk/phoenics/d\\_polis/d\\_lecs/general/maths.htm](http://www.cham.co.uk/phoenics/d_polis/d_lecs/general/maths.htm), 2003.
4. Harlow F. H. and Nakayama P. I., "Transport of turbulence energy decay rate", Los Alamos Science Laboratory, LA-3854, Univ. of California, USA, 1968.

### **Nomenclature**

$U$	vector velocity, u, v & w
$\rho$	density, kg/m <sup>3</sup>
$\phi$	general flow variables such as velocity & temperature
$\Gamma_{\phi}$	diffusive exchange coefficient for flow variable
$S_{\phi}$	source term

### **Subscript**

j	x, y & z
---	----------

### **Appendix**

Source or input file (Q1 file)