Analyze Air Flow Uniformity over the Condenser Face with 'V' Arrangement for Air Cooled Water Chiller: - CFD Simulation.

Udhhav Nimbalkar[1], Nitin Gulhane[2], Anil kumbhar[3],

[1]*Assistant professor, Department of Mechanical Engg, VJTI Mumbai.,nimbalkaruddhav@gmail.com*
[2]*Associate Professor, Department of Mechanical Engg, VJTI Mumbai, npgulhane@vjti.org.in*
[3]*Assistant General Manager, EM&RBG division, Voltas Thane, akumbhar@voltas.com*

ABSTRACT

This paper represents the study of air flow uniformity over the condenser face which is used in large air cooled water chiller. This study investigates performance as well as efficiency enhancement, for a specified chiller model. With the help of CFD tool investigate the required unknown parameter, and optimize the design .In this research work, one 'V' shape condenser having micro channel coil model for CFD analysis, to find out how flow is distributed along the condenser face .Find out different shapes of condenser coils based upon improve maximum heat transfer coefficient so that our chiller performance will be improve. Basic intension is the how to maximize the flow distribution along the condenser face so that we will reduce the size of condenser, air cooled chiller having capacity 100-300 TR and result shows the how air velocity is to be distributed 1m/s to 6m/s along the condenser face as well as velocity distribution in interior. General conclusion related to the factors affected for the uniform air distribution and enhancement of heat transfer coefficient so that optimization of chiller unit take place.

Keywords*:* Computational Fluid Dynamics, Air cooled chiller, Microchannel coil, COP.

---

1. Introduction

In air conditioning systems, chilled water is typically distributed to heat exchangers, or coils, in air handling units or other types of terminal devices which cool the air in their respective space(s), and then the water is re-circulated back to the chiller to be cooled again. Chilled water temperatures can range from 35 to 45 °F (2 to 7 °C), depending upon application requirements.

In industrial application, chilled water or other liquid from the chiller is pumped through process or laboratory equipment. Industrial chillers are used for controlled cooling of products, mechanisms and factory machinery in a wide range of industries. They are often used in the plastic industry in injection and blow molding, metal working cutting oils, welding equipment, die-casting and machine tooling, chemical processing, pharmaceutical formulation, food and beverage processing, paper and cement processing, vacuum systems, X-ray diffraction, power supplies and power generation stations, analytical equipment, semiconductors, compressed air and gas cooling. They are also used to cool high-heat specialized items such as MRI machines and lasers, and in hospitals, hotels and campuses.

Chillers for industrial applications can be centralized, where a single chiller serves multiple cooling needs, or decentralized where each application or machine has its own chiller. Each approach has its advantages. It is also possible to have a combination of both centralized and decentralized chillers, especially if the cooling requirements are the same for some applications or points of use, but not all.

Decentralized chillers are usually small in size and cooling capacity, usually from 0.2 tons to 10 tons. Centralized chillers generally have capacities ranging from ten tons to hundreds or thousands of tons.

Air-cooled water chillers are widely used to cool water or secondary coolants in air-conditioning and refrigeration applications. Air-cooled water chillers are mainly equipped with a compressor, a condenser, a throttling device, and an evaporator, among which the condenser comprises several finned-tube heat exchangers (hereafter called condenser coils) and fans. The arrangement of the condenser coils and the fans forms a VV-shaped configuration as shown in Fig. 1. In general, the flow velocity of air passing a condenser coil is assumed to be constant and uniform in the calculation of the heat transfer rate when designing condenser coils. However, in practical applications the fans are placed above the condenser coils in the VV-shaped configuration, causing uneven upper and lower airflow through the coils as well as considerable differences between flow velocities in the inner and outer coils. The actual heat transfer rate thus becomes lower than the design value, raising the condensing temperature and

ISSN: 2321-8169
457 – 463

reducing the energy performance of the water chiller. Therefore, the investigation of methods to achieve uniform airflow and velocity over the coil surface and enhance equipment performance is an important research topic. [2]

Fig1: V configuration condenser coils

Above fig. shows V configuration arrangement, in that exhaust fans are locate on the top surface of a chiller body .Due to pressure jump create with the help of exhaust fan ambient air get suck over the condenser face which containing micro channel coil. Which produce the required convective coefficient for overcome the condenser load. Basic aim of an analysis is in theoretical calculation we assume the constant convective heat transfer coefficient for design calculation of condenser, but in actual practice distribution of air velocity on condenser face is not uniform. Because of that we get deviation in design parameter. To overcome this problem with the help of computational fluid dynamics tool analyze the required parameter. Average air velocity was determined for these parts, using the airflow distribution obtained in the above CFD computer simulation. The average air velocity was then used to calculate the heat transfer rates for each part on the coils. The heat transfer for each of these regions can then be calculated and the total heat transfer can be determined by summing the heat transfer of each region. [1]

$$Qt = \sum Qi$$

Qt = total heat transfer.

Qi = heat transfer in region

Heat transfer in each region can be calculated by .

Qi = ∈ Cmin (Thi – Tci) .

Where ε is the heat transfer effectiveness; Cmin is the air side heat capacity rate (W/K); Thi and Tci are the inlet temperatures of hot (refrigerant) and cold (air) fluids, respectively. As the refrigerant side heat capacity rate, Cmax is much larger than Cmin

1.1 Basic equation used in for CFD analysis

Continuity equation

$$\frac{\partial \rho}{\partial t} + div(\rho u) = 0 \qquad \text{Eq.1}$$

Momentum equation

$$\frac{\partial}{\partial t}(\rho u) + div(\rho u \upsilon) = \frac{-\partial \rho}{\partial x} + div(\mu \ grad \ u) + S_{Mx} \qquad \text{Eq.2}$$

$$\frac{\partial}{\partial t}(\rho v) + div(\rho v \upsilon) = \frac{-\partial \rho}{\partial y} + div(\mu \ grad \ v) + S_{My} \qquad \text{Eq.3}$$

$$\frac{\partial}{\partial t}(\rho w) + div(\rho w \upsilon) = \frac{-\partial \rho}{\partial z} + div(\mu \ grad \ w) + S_{Mz} \qquad \text{Eq.4}$$

In above equations, S_M and ϕ are defined as momentum source and dissipation function respectively.

1.1 Selection parameters of chiller

Important specifications to consider when searching for industrial chillers include the total life cycle cost, the power source, chiller IP rating, chiller cooling capacity, evaporator capacity, evaporator material, evaporator type, condenser material, condenser capacity, ambient temperature, motor fan type, noise level, internal piping materials, number of compressors, type of compressor, number of fridge circuits, coolant requirements, fluid discharge temperature, and COP (the ratio between the cooling capacity in RT to the energy consumed by the whole chiller in KW). For medium to large chillers this should range from 3.5-7.0 with higher values meaning higher efficiency. Chiller efficiency is often specified in kilowatts per refrigeration ton (kW/RT).

2. Porosity resistance

Our analysis purpose we assume condenser face as a porous medium, because condenser having micro channel coils. For that purpose in CFD analysis of porous media estimate the parameter with the help of Ergun equation. [4]

Total pressure drop = Viscous loss + inertial loss.

$$\frac{dp}{dl} = \left[\frac{150 \times \mu \times (1-e)^2}{phi^2 \times D^2 \times e^3}\right] \times v + [\frac{1.75 \times \rho \times (1-e)}{phi \times D \times e^3}] \times v^2 \qquad \text{Eq.5}$$

The first term is viscous loss and second term is internal loss. Compare above equation with momentum sink equation for fluent analysis.

$$\frac{dp}{dl} = R_v \times \mu \times v + \left(\frac{R_i}{2}\right) \times \rho \times v^2 \qquad \text{Eq.6}$$

This gives the values for R_v and R_i

$$R_v = \left[\frac{150 \times (1-e)^2}{phi^2 \times D^2 \times e^3}\right]$$

$$R_i = [\frac{1.75 \times 2 \times (1-e)}{phi \times D \times e^3}]$$

Using above equation and geometrical data we calculate the viscous and internal resistances

$$R_v = 1.39 \times 10^8 \quad (m^{-2})$$

$$R_i = 154.35 \quad (m^{-1})$$

Use above two values use in fluent analysis for porous media approach.

2.1 Modeling

Assumption:

- The analysis will be carried out in steady state operating condition.
- No fan will be modeled, only uniform air flow rate is assumed at fan outlet.
- No ambient air will be considered after fan outlet.
- Gravitational effect will be considered.

Preprocessing tools like model generation, mesh generation, defined boundary condition in ANSYS GAMBIT. Fig2 shows actual model view in gambit. In that condenser face is directly consider as a porous medium for analysis purpose. Prepare domain which contain from condenser face to exhaust fan which located in upper portion. Dimension of a model is given below.

Height	1.1176m
Length	1.9558m
Height	1.1176m
Length	1.9558m
Width	0.066m
Fan diameter	0.70m

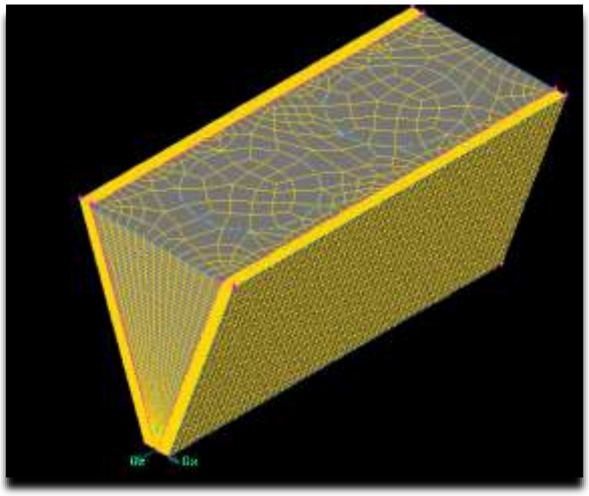
Table1: Physical dimensions.

Due to symmetric arrangement we consider only one coil for analysis. Above diagram contain three volumes in that two having condenser and middle portion for interior, Upper two circular sections for exhaust fan. Mesh type used is Hex/Wedge cooper in gambit. Boundary conditions are as follow.

Sr.no.	Boundary Conditions
1	Pressure inlet
2	Exhaust fan
3	Wall
4	Porous medium
5	Air-fluid
6	Aluminium-material for condenser.

Table2: Boundary condition

Above boundary condition pressure inlet is considered as absolute pressure 101325Pa.Exhaust fan having 12000cfm capacity & pressure gauge reading is 10mm of water. All walls are stationary. After meshing whole model is import into fluent



No of element created in each of condenser volume is 125000 after meshing.

3. **Results and discussion**

This study used air cooled chiller for full-scale experimention in order to determine the actual heat transfer capacity of condenser, and COP of the chiller in actual operation. These data is used to verify the accuracy of the simulation result.

Actual convective heat transfer coefficient (h_o) shell side given by a following corealation [1]

$$j = St \times Pr^{\frac{2}{3}} = \frac{h_o \times Pr^{\frac{2}{3}}}{G \times C_p} = .0014 + 0.2618Re^{-0.4}(\frac{Ao}{At})^{-0.15}$$

Where, St is Stanton number, Pr is prandtl number and Re is Reynolds number which depend upon air flow velocity and tube spacing.

Velocity distribution along the condenser face varies between in the range of o to 6 m/s which is shown in fig5. As you go toward the exhaust fans velocity of air increase due to increase in pressure drop.[4]

Following graph shows the velocity distribution along the height of a coil in different location . It shows results of velocity along the condeser coil in three different location .

ISSN: 2321-8169
457 – 463

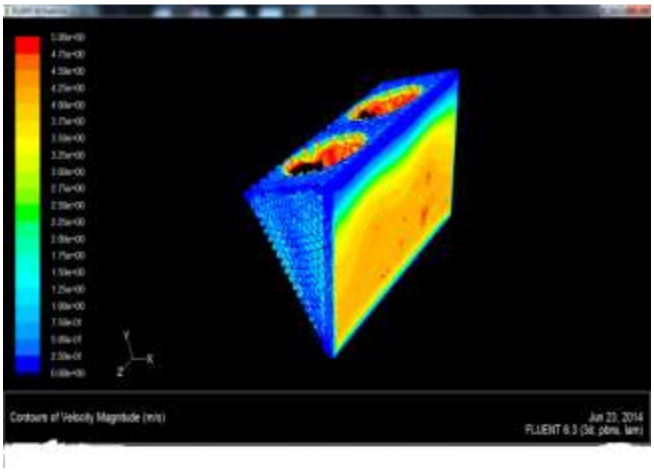
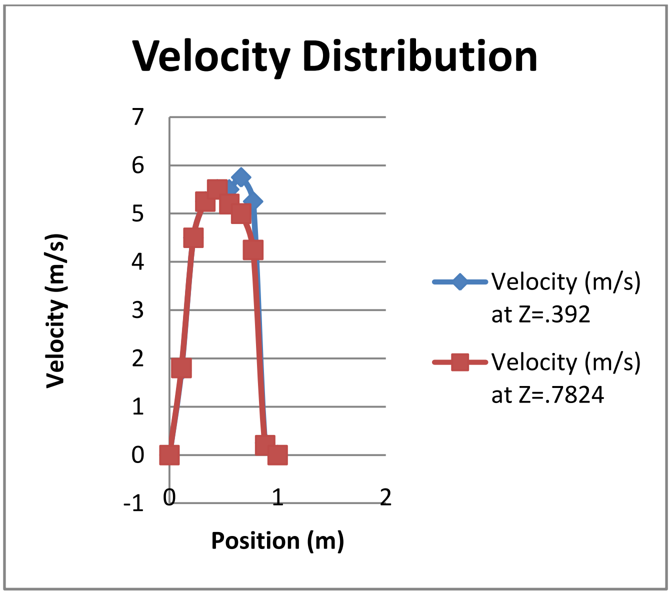


Fig3: velocity contour along the face

Following graph shows the velocity distribution along the height of a coil in different location . It shows results of velocity along the condeser coil in three different location .



Graph1: position Vs velocity magnitude.

As we go from section one to section three velocity of the air increase slowly depend upon pressure difference created by the exhaust, Maximum velocity along face is 6(m/s) which is the nearer to the exhaust fan. It clearly gives the velocity values required position of a coil and with the help of thease value we find out the actual heat transfer of condenser as well as capacity of condenser.

After that fig6 gives the velocity distribution in interior part of a condenser,means area enclosed by two condenser coils. For that purpose we create two iso surface along the y direction exact below the exhaust fan, it gives the velocity sections along the height.

ISSN: 2321-8169
457 – 463

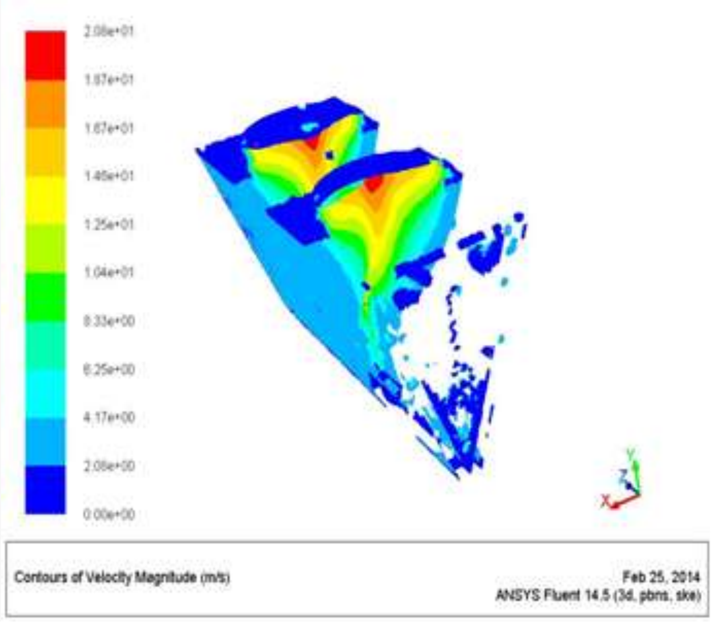


Fig4: velocity contour in interior.

Above fig shows maximum velocity at the exhaust fan is 20(m/s),it shows how velocity is distributed along the interior part. With the help of above all the results find out side convective heat transfer coefficient (h_o) and calculate the maximum possible heat transfer . Our analysis aim is to be optimization of shape of a condenser coil. For air cooled chiller package create more turbulance along the face of condenser so that increase the heat transfer rate of condenser.

3.1 Result validation

For validation purpose we take five different reading along the height of coil with respect to five different locations in Z direction

Sr. no. Z= 0.392	Theoretical Results of velocity (m/s)	Experimental Results (m//s)
1	3.125	2.34
2	4.875	3.73
3	5.3	5.2
4	5.5	6.11
5	5.625	6.82

Table 2 Result validation at Z= 0.392

4 CONCLUSION

The objective of this study was to construct a parameter analysis model in which the VV configuration condenser coil analyzed, and to developed mathematical model models of the overall system and individual component of an air cooled water chiller. With the help of analysis we can analyze the different between actual and experimental velocity reading and analyze difference between theoretical COP and actual experimental COP. Also compare the different coils configuration such as WW, inverted VV, and select the best design and optimize the best air cooled condenser coil shape.

ISSN: 2321-8169
457 – 463

Nomenclature

∂	density (kg/m^3)
t	time (s)
Cp	heat capacity (J/kg K)
u,v, w	velocities in x, y and z direction respectively (m/s)
K	thermal Conductivity (W/m K)
P	pressure (N/m^2)
x, y, z	cartesian Coordinates (m)
m	mass flow rate (kg/s)
h	heat transfer coefficient (W/m^{2}K)
R_v	viscous resistance
R_i	internal resistance
e	porosity of the medium
D	diameter of the particle making medium
v	average velocity
phi	sphericity (assume 0.75)
T	temperature (K)
μ	dynamic viscosity (Ns/m^2)
υ	velocity vector (m/s)

REFERENCES

1. Tzong-Shing Lee*,1, Wu-Chieh Wu, Sheng-Kai Wang. Improved energy performance of air- cooled water chillers with innovative condenser coil configurations e Part I: CFD.
2. Tzong-Shing Lee*, Wu-Chieh Wu, Jin-Cheng Jiang. Improved energy performance of air-cooled water chillers with innovative condenser coil configurations e Part II: Experimental validation.
3. Wu-Chies Wu, Tzong-Shing Lee* and Chich-Hsiang Chang. Energy and Exergy Analysis For Improving The Energy Performance Of Air Cooled Liquid Chillers by Different Condensing-Coil Configurations.
4. ANSYS; FLUENT Tutorial Guide ; Release 14 (November 2011)