International Journal of Research
Available at https://edupediapublications.org/journals
p-ISSN: 2348-6848
e-ISSN: 2348-795X
Volume 02 Issue 11
November 2015

Computational Fluid Dynamics Analysis of Two Dimensional Condenser

Mohit Kumar, Shubham Singhmar, Shubham Gupta
Student, Department Of Mechanical Engineering, Chandigarh University, India
Student, Department Of Mechanical Engineering, Chandigarh University, India
Student, Department Of Mechanical Engineering, Chandigarh Polytechnic College, India

Abstract

Heat pipe is a thermodynamic device of very high thermal conductance. In a heat pipe the working fluid experiences phase change both in the evaporator and condenser sections. In this paper modeling a heat exchanger/condenser is a more complex problem with useful applications. A two-dimensional approach was initially used to gain insight into the problem before moving onto a full three-dimensional condenser with numerous cooling tubes and baffles. The central tube prevents the air from traveling directly from the input pipe to the output pipe, instead forcing the air to circulate and spend more time being cooled before exiting.

Keyword:-CFD, Heat Transfer, Condenser, Fluent

Introduction

Computational fluid dynamics (CFD) is the use of computers and numerical methods to solve Problems involving fluid flow. CFD has been successfully applied in many areas of fluid mechanics. These include aerodynamics of cars and aircraft, hydrodynamics of ships, flow through pumps and turbines, combustion and heat transfer chemical engineering. Applications in civil engineering include wind loading, vibration of structures, wind and wave energy, ventilation, fire, explosion hazards, dispersion of pollution, wave loading on coastal and offshore structures, hydraulic structures such as weirs and spillways, sediment transport. More specialist CFD applications include ocean currents, weather forecasting, plasma physics, blood flow and heat transfer around electronic circuitry.

Basic Principles of CFD

The approximation of a continuously-varying quantity in terms of values at a finite number of points is called discretisation.

The fundamental elements of any CFD simulation are:
(1) The flow field is discredited; i.e. field variables (, u, v, w, p …) are approximated by their values at a finite number of nodes.
(2) The equations of motion are discretised (approximated in terms of values at nodes):

Control-volume or differential equations	→	algebraic equations
(Continuous)		(Discrete)

The main stages in a CFD simulation are:

Pre-processing:
– Formulation of the problem (governing equations and boundary conditions);
– Construction of a computational mesh (set of control volumes).

Solving:
– Discretisation of the governing equations;
– Solution of the resulting algebraic equations.
Post-processing:
– Visualization (graphs and plots of the solution);
– Analysis of results (calculation of derived quantities: forces, flow rates ...)

Forms of the Governing Fluid-Flow Equations

The equations governing fluid motion are based on fundamental physical principles:
Mass: change of mass = 0
Momentum: change of momentum = force × time
Energy: change of energy = work + heat

In fluid flow these are usually expressed as rate equations; i.e. *rate of change* = …

When applied to a fluid continuum these *conservation* principles may be expressed Mathematically as either:
Integral (i.e. *control-volume*) equations;
Differential equations.

Integral (Control-Volume) Approach

This considers how the **total amount** of some physical quantity (mass, momentum, energy,) is changed within a specified region of space (*control volume*).
For an arbitrary control volume the balance of a physical quantity over an interval of time is
Change = amount in − amount out + amount created
In fluid mechanics this is usually expressed in **rate** form by dividing by the time interval (and transferring the net amount passing through the boundary to the LHS of the equation):

$$\begin{pmatrix}\text{RATE OF CHANGE}\\ \textit{inside } V\end{pmatrix} + \begin{pmatrix}\text{NET FLUX}\\ \textit{out of boundary}\end{pmatrix} = \begin{pmatrix}\text{SOURCE}\\ \textit{inside } V\end{pmatrix}$$

The *flux*, or rate of transport across a surface, is composed of:
Advection – movement with the fluid flow;
Diffusion – net transport by random molecular or turbulent motion.

$$\begin{pmatrix}\text{RATE OF CHANGE}\\ \textit{inside } V\end{pmatrix} + \begin{pmatrix}\text{ADVECTION} + \text{DIFFUSION}\\ \textit{through boundary of } V\end{pmatrix} = \begin{pmatrix}\text{SOURCE}\\ \textit{inside } V\end{pmatrix}$$

The important point is that this is a **single, generic equation**, irrespective of whether the

International Journal of Research
Available at https://edupediapublications.org/journals
p-ISSN: 2348-6848
e-ISSN: 2348-795X
Volume 02 Issue 11
November 2015

physical quantity concerned is mass, momentum, chemical content, etc. Thus, instead of Dealing with lots of different equations we can consider the numerical solution of a generic *Scalar-transport equation*

Differential Equations

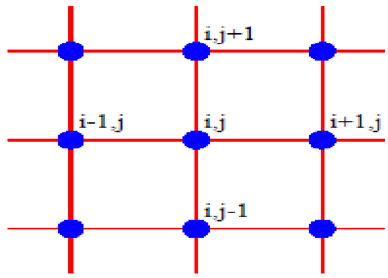
In regions without shocks, interfaces or other discontinuities, the fluid-flow equations can also be written in equivalent **differential** forms. These describe what is going on at a **point** rather than over a whole control volume. Mathematically, they can be derived by making the control volumes infinitesimally small.

The Main Discretization Methods

(1) Finite-Difference Method

Discretize the governing **differential** equations; e.g.

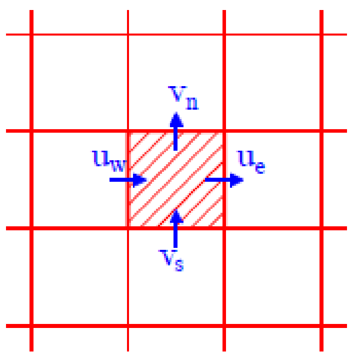
$$0 \;=\; \frac{\partial u}{\partial x}+\frac{\partial v}{\partial y} \;\approx\; \frac{u_{i+1,j}-u_{i-1,j}}{2\Delta x}+\frac{v_{i,j+1}-v_{i,j-1}}{2\Delta y}$$



(ii) Finite-Volume Method

Discretize the governing **integral** or **control-volume** equations; e.g.

$$net\ mass\ outflow \;=\; (\rho uA)_e-(\rho uA)_w+(\rho vA)_n-(\rho vA)_s \;=\; 0$$



(iii) Finite-Element Method

Express the solution as a weighted sum of *shape functions* S (**x**); e.g. for velocity:

p-ISSN: 2348-6848
e-ISSN: 2348-795X
Volume 02 Issue 11
November 2015

$$u(\mathbf{x}) = \sum u_\alpha S_\alpha(\mathbf{x})$$

Substitute into some form of the governing equations and solve for the coefficients u_α

Boundary Conditions: The large exterior circle has a radius of 20 cm with the interior "tubes" having radii of 3.7 cm or 2.5 cm. The input and output "pipes" have a diameter of 5 cm. Air enters at the top with a temperature of 375 K and a downwards velocity of 10 m/s and flows out the bottom. The interior tubes are held at a temperature of 310 K.

Case 1.

1. Geometry

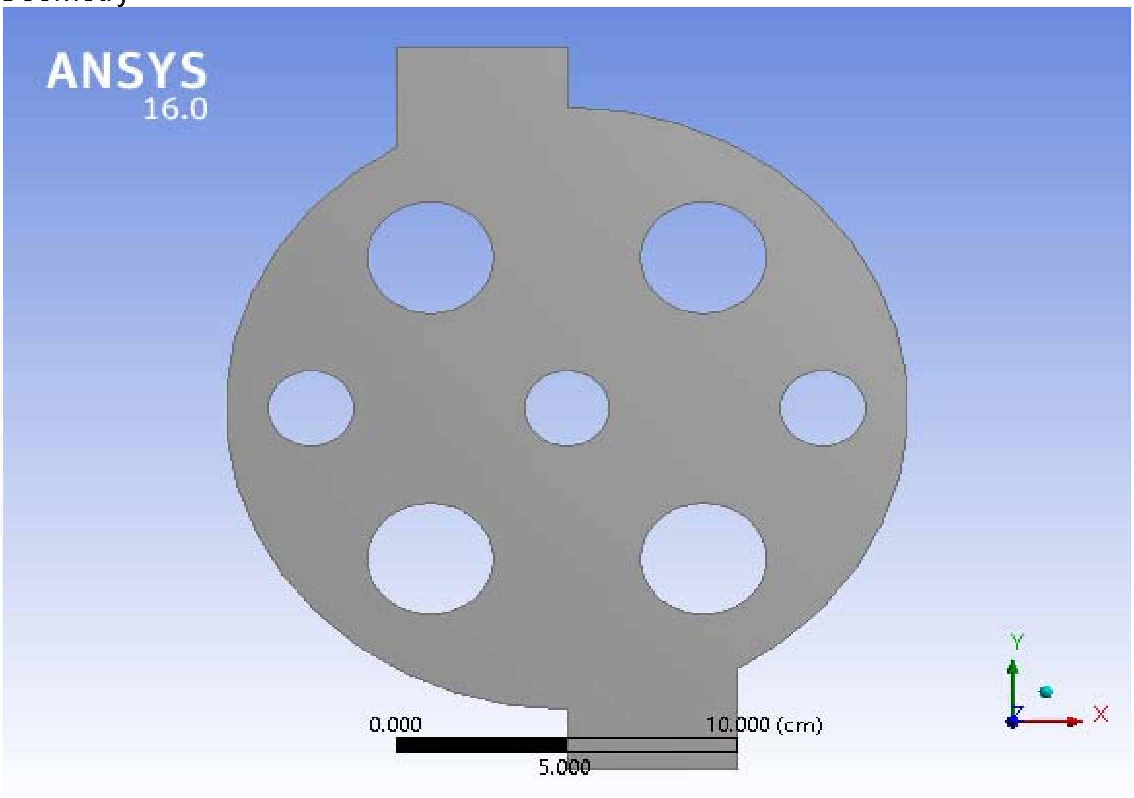


Figure 1.Geometry Case 1

2. Meshing & Name Selection

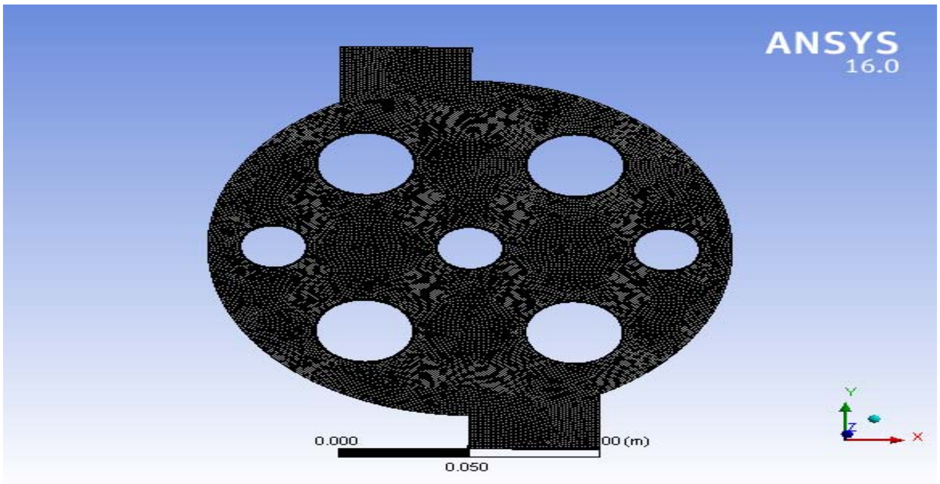


Fig2: Meshing

In Meshing body size is provided 1mm. Inflation is provided on inlet, outlet or tube. Name created in meshing.

3. Boundary Condition

In Ansys Setup provide boundary Condition The large exterior circle has a radius of 20 cm with the interior "tubes" having radii of 3.7 cm or 2.5 cm. The input and output "pipes" have a diameter of 5 cm. Air enters at the top with a temperature of 375 K and a downwards velocity of 10 m/s and flows out the bottom. The interior tubes are held at a temperature of 310 K.

4. Result

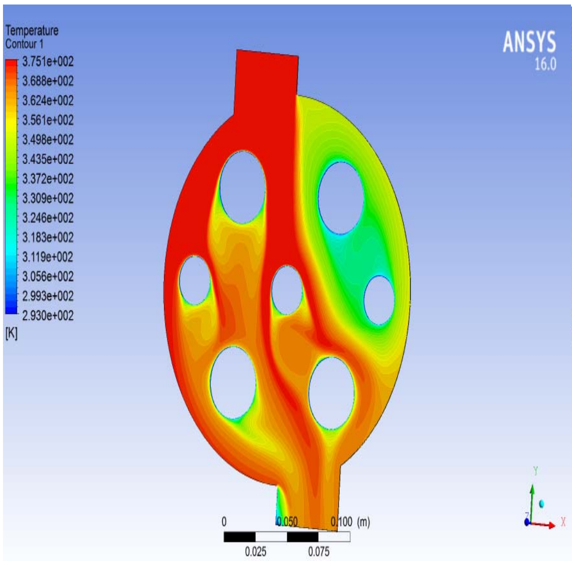


Fig3: Contours of temperature

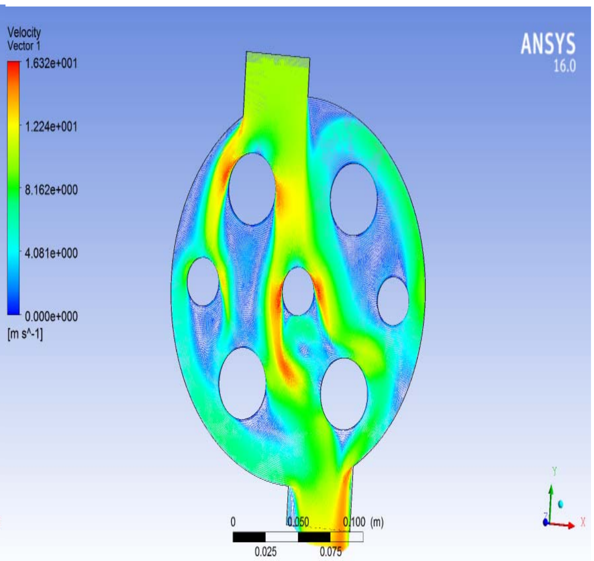


Fig4: Velocity vectors

International Journal of Research
Available at https://edupediapublications.org/journals
p-ISSN: 2348-6848
e-ISSN: 2348-795X
Volume 02 Issue 11
November 2015

Case 2.

1. Geometry

2. Meshing

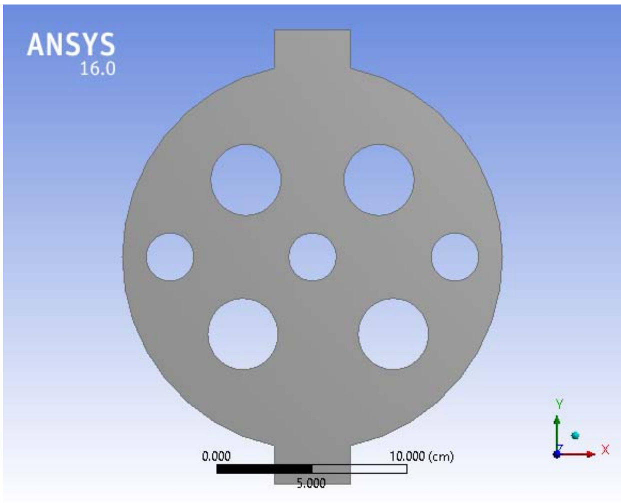


Fig4: Geometry Case2

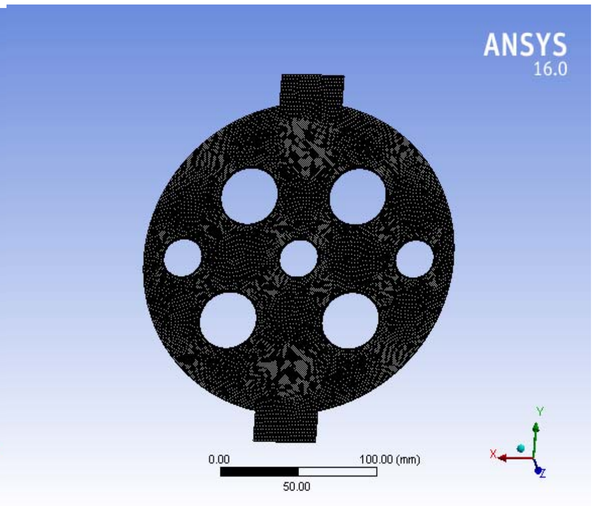


Fig5: Meshing

4. Result

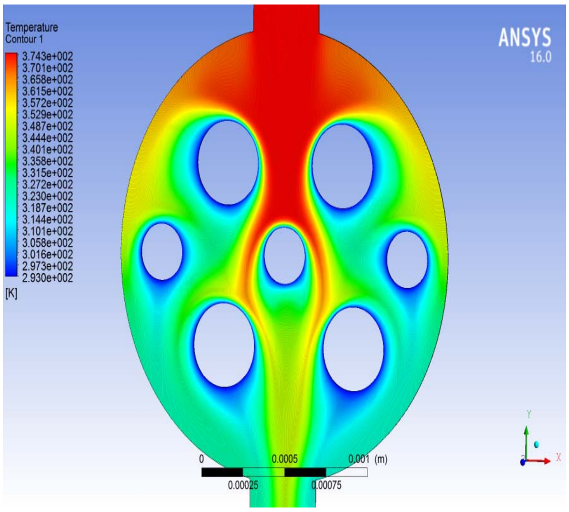


Fig6: Contours of temperature

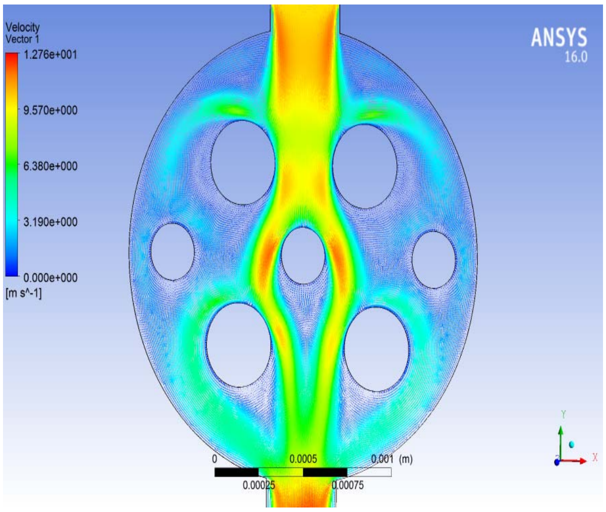


Fig7: Velocity vectors

Conclusion

The velocity vector field shows that the central tube is important in diverting the flow of the hot air (Fig 4). Additionally, areas of circulation can be seen and these correspond to the cooler areas in the temperature contours (Fig 3). There is significant cooling of the air as it passes through the condenser. But Case 2 symmetry geometry the cooled area in the temperature contour (Fig6) is better than case1.