Nanotechnology Perceptions
ISSN 1660-6795
www.nano-ntp.com

Design Optimization Several Computational Fluid Dynamic Investigations Carried of Various Warm Interchanger

Prabhat Kaushal[1], Dr. Manish Gangil[2]

[1]PhD Research Scholar1 RKDF University, Airport Road, Gandhi Nagar, India
[2]Professor, RKDF University, Airport Road, Gandhi Nagar, India
Email: prabhatkaushal87@gmail.com

Warm interchanger is one of the key aspects of machineries, devices and industrial processes for maintaining their functionality and also for achieving better product quality. Hence, heat exchangers of different types and sizes are used in these applications with the purpose of removing the extra process/device heat to maintain the desirable working temperatures. However, the size of a heat exchanger is a major consideration for any type of process/device as it decides the space requirements (i.e., the size) of the machine/device or the processing plant. At first, this study aims to investigate the design procedure of a heat exchanger theoretically and then its performance will be analyzed and optimized using computational fluid dynamics. Afterwards, a computational model of the same heat exchanger was implemented with ANSYS and then this model was extended to six different models by altering its key design parameters for the optimization purposes. Eventually, these models were used to analyze the warm interchanger behavior, mass flow rates, pressures drops, flow velocities and vortices of shell and tube flows inside the heat exchanger. The axial pressure drops showed positive correlations with both the overall warm interchanger coefficient and pumping power demand.

Keywords: Computational Fluid Dynamic (CFD), Physical model, computational model. ANSYS Fluent.

1. Introduction

Most of the literature work published earlier in this field of warm interchanger enhancement in the automotive radiators had emphasized several techniques that has a considerable increase in the warm interchanger rate. However most of the experiments and analysis done earlier are performed in circular tubes and fair results are obtained with different boundary conditions. From the literature survey, it was found that warm interchanger enhancement studies were not reported largely in rectangular ducts. The reason to choose a non-circular ducts is that because of the pressure drop limitations that arises in many warm interchanger applications in circular cross sections. The reason that led to investigate fluid made to flow through the non-circular ducts is that sometimes the need for noncircular ducts arises in many warm

interchanger applications particularly in compact heat exchangers. Also the warm interchanger rates through the noncircular ducts (triangle, square, rectangle etc.) are smaller than that of circular tubes due to less pressure drop.

Computational investigations were carried out for the present experimental domain and the results are compared. Suitable solver setting, turbulence model was identified from the literature and the boundary conditions for the simulations were taken from the present experiment.

Numerical investigations were based on 3D steady state conjugate heat transfer analysis to infer flow and warm interchanger characteristics of the absorber fluid using the FVM based code Ansys Fluent ® 15.0. The objective of the CFD analysis was to validate the computational methodology adopted with that of the experimental results.

2. Physical Model

Computations are planned for conducting tests on a simplified and scaled up radiator tube, for easy provision and access of the instrumentation. Temperature values measured at strategic locations on the radiator tube and at the fluid inlet and outlet will be reported. The comparison of performance of the radiator tubes with different winglet turbulator configurations will be carried out and reported. The reason that the test section chosen is rectangular duct because its warm interchanger surface area is very less when compared to the conventionally used ducts in the radiators, and it also seems to be more compact and efficient. As any savings in the material required for the manufacture of the radiators will result in a huge financial savings which in turn causes huge profit for the manufacturers.

3. Computational Model

Computational Fluid Dynamics (CFD) is the science of predicting fluid flow, heat transfer, mass transfer, chemical reactions, and related phenomena by solving the mathematical equations which govern these processes using a numerical process. The result of CFD analyses is relevant engineering data used in conceptual studies of new designs, detailed product development, troubleshooting and redesign. CFD analysis complements testing and experimentation and reduces the total effort required in the laboratory. Analysis begins with a mathematical model of a physical problem. Conservation of matter, momentum, and energy must be satisfied throughout the region of interest. In general, the fluid properties are modeled empirically. Simplifying assumptions are made in order to make the problem tractable (e.g., steady-state, incompressible, inviscid, and two-dimensional).

The user needs to provide appropriate initial and boundary conditions for the problem.

4. Computational Methodology

Conservation laws of a continuum medium involve vector and tensor quantities. Following three different types of notations are usually employed in continuum mechanics:

Dyadic or vector notation Expanded or component form Cartesian tensor notation

Most of the primary variable solution techniques for incompressible flows are built on either an explicit or an implicit time marching scheme. Majority of these methods can be classified as projection methods which first construct a velocity field that does not satisfy continuity equation, and then correct it by subtracting something (e.g. a pressure gradient) to enforce continuity. A typical numerical simulation algorithm for Navier-Stokes equations consists of two separate discretization steps:

i. Spatial discretization and

ii. Temporal discretization.

The spatial discretization using finite difference, finite volume or finite element method results in a system of semi-discrete coupled non-linear ordinary equations in time which are solved using an appropriate time marching scheme (implicit or explicit). Further, at each time step, we may need to solve a set of linear or non-linear algebraic equations depending on the choice of time integration algorithm. In what follows in this lecture, we tacitly assume that Navier-Stokes equations have already been discretized in space by the analyst using her/his favorite discretization method (namely FDM, FVM or FEM), and we present the overall solution algorithm following this step. The scalar values) are stored at the cell centres. The face value of ( ) is required for the convection terms and it must be interpolated from the cell center values. This is accomplished using an upwind scheme.

The face value of) is derived from the quantities in the cell by upstream or upwind, scheme related to the direction of normal velocity. To obtain first order accuracy the quantities at all cell faces are determined by assuming the cell-center values of field variable which represent a cell- average throughout the entire cell. In order to get second order accuracy, the cell faces are computed using a multidimensional linear reconstruction approach. In this approach, the higher-order accuracy is achieved at cell faces through a Taylor series expansion of cell-centres solution about the cell centroid.

5. Steps in Basic Numeric Model

In the numerical model, a control volume formulation is used. The above equations are integrated over each control volume to obtain a set of discretized linear algebraic equations. The iterative procedure is used for getting the discretized transport equations because the equations are coupled and the convection terms in momentum equations are non-linear. The pressure field does not have any transport equation, but the pressure source term in the momentum equation has to be calculated. The accurate values of pressure field would give exact values of velocities that satisfy mass continuity. The pressure field can be obtained indirectly from the continuity equation, since the calculated velocities from the momentum equation have to satisfy mass continuity. The solution procedure starts with the initial guessed fields for calculated variables (velocity, pressure, turbulent, quantities, temperature and concentration). Initially the guessed pressure field p* is used to calculate velocity components that are denoted u*, v* and w*. Then, the pressure field has to be corrected in a way that the velocity field progresses towards satisfying the continuity equation. The correct pressure p is linked to the pressure correction p' and it is given in the following form

$p = p^* + p'$

In the similar way the velocities can be defined as $u = u^* + u'$

$v = v^* + v'$ $w = w^* + w'$

The velocity correction equation has the same form as the final discretization equation. However, the influence of the neighbor cells is neglected since the flow field is involved with velocity and pressure correction for the particular cell. Therefore, the velocity correction equation has only the source term based on the pressure correction. Solution of the pressure correction gives values for the correct pressure field p. Then the velocity correction equation and velocity components are solved. After pressure and velocity fields are calculated, the other scalar variables are also calculated. The iterations continue with the calculated values as new guesses for variables. This procedure is repeated until the convergent solution is reached.

The main steps in the SIMPLE algorithm are

i) Guess a pressure field.

ii) Solve the momentum equations using this pressure field, thus obtaining the velocities which satisfy momentum.

iii) Construct continuity errors for each cell: inflow - outflow

iv) Solve the pressure correction equation. The coefficients are d(vel) / d (p), and the sources are the continuity errors.

v) Adjust the pressure and velocity fields. Obtain velocities, which satisfy continuity, but not momentum.

vi) Go back to step 2 and repeat with the new pressure field. Repeat until continuity and momentum errors are acceptably small.

6. Steps in Computational Fluid Dynamics (CFD)

GAMBIT is a software package designed to help analysts and designers in geometry and mesh generation. GAMBIT's single interface for geometry creation and meshing brings together most of Fluent's pre- processing technologies in one environment. GAMBIT can dramatically reduce pre-processing times for many applications. Most models can be built directly within GAMBIT's solid geometry modeler, or imported from any major CAD/CAE system. Using a virtual geometry overlay and advanced clean-up tools, imported geometries are quickly converted into suitable flow domains.

A comprehensive set of highly automated and size function driven meshing tools ensure that the best mesh can be generated, whether structured, unstructured, or hybrid. When it comes to meshing, the choices before any engineering analysis are to choose between a structured mesh and an unstructured mesh. Researchers recommends that the structured mesh is less memory consuming, and the extraction and analysis of the data are more direct, and allow more straightforward analysis with respect to experimental measurements.

Researchers emphasized that structured grids will, compared to unstructured grids, often be

more efficient in terms of accuracy, CPU time and memory requirement. Using hexahedral cells, the first node walls distance parameter y+ can be controlled close to unity or less, which is necessary to capture the flow features as accurately as possible. Fine near wall meshing is provided to the wall surfaces in contact with fluid, using a boundary layer meshing technique in the pre-processor. The near wall mesh is controlled by attaching a boundary layer with the first node being placed as close as possible in the direction normal to the wall. This is decided by approximately estimating the y+ value to be about unity. The mesh is block structured and the elements are hexahedral.

Pre-processing tool Gambit 2.4 was used to generate HX dominant finite volume elements throughout the computational domain. Finer near wall meshing is developed using the boundary layer mesh feature of the pre- processor with the first node at 0.1mm and with a default growth factor of 1.1 for 10 layers, to capture the flow and accurate happening of thermal features close to the wall. Hybrid meshing with a combination of hexahedral and tetrahedral mesh elements was used throughout the computational domain.

FLUENT is a state-of-the-art computer program for modeling fluid flow and warm interchangerin complex geometries. It provides complete mesh flexibility, solving our flow problems with unstructured meshes that can be generated about complex geometries with relative ease. Supported mesh types include 2D triangular/quadrilateral, 3D tetrahedral / hexahedral / pyramid / wedge and hybrid meshes. FLUENT also allows refining or coarsening the grid based on the flow solution. FLUENT is written in the C computer language and makes full use of the flexibility and power offered by the language. Consequently, true dynamic memory allocation, efficient data structures and flexible solver control are all made possible.

All functions required to compute a solution and display the results are accessible in FLUENT through an interactive, menu-driven interface.

The basic procedural steps to obtain CFD simulated results are shown below.

1. Creating the model geometry and mesh-using GAMBIT.
2. Start the appropriate solver 3D modeling in FLUENT.
3. Import the grid into FLUENT.
4. Check the grid.
5. Select the solver formulation.
6. Choose the basic equations to be solved
7. Specification of material properties.
8. Specification of boundary conditions.
9. Adjusting the solution control parameters.
10. Initializing the flow field.
11. Start run in solver
12. Examining the results.

13. Saving the results.

Based on the mass flow rate and geometry, the Reynolds numbers are found to fall well below the critical value and hence the flow can be treated as laminar. More details of the governing equations will be presented in a subsequent section of this chapter. In the present study the solver parameters are steady, implicit, absolute velocity formulation with Green- Gauss cell based gradient options. SIMPLE scheme was used for the pressure velocity coupling and second order up-winding was chosen for discretization of the governing equations.

CFD is nothing but solving the fundamental governing equations of fluid dynamics which are essentially mathematical models of conservation laws of physics.

a. Conservation of Mass, Momentum and Energy

Assuming a fluid to be a continuum, these conservation laws are

1. Conservation of mass
2. Conservation of momentum (Newton's second law)
3. Conservation of energy (first law of thermodynamics)

These conservation laws are supplemented with constitutive relations (e.g. stress-strain rate relation, heat diffusion law, etc.) for a specific material. Numerical simulation of the flow problems involves solution of Navier-Stokes equation. This vector equation is essentially a set of three coupled partial differential equations, and must be solved along with the continuity equation and energy equation. All these equations are coupled PDEs, and their collection is often referred to as Navier-Stokes equations in CFD literature. One remarkable feature of these equations is the similarity of their form: each of these equations can be recast in the form of the generalized transport equation which contains a time derivative term, a convective term, a diffusive terms and a source term.

For compressible flows, the continuity equation, momentum equation and energy equation represent the transport equations for density, velocity and temperature respectively. Therefore, the discretization methods can be applied to each equation, and collection of the resulting discretized nonlinear equations can have solved using a sequential iterative scheme. Thus, algorithm for numerical solution of unsteady subsonic compressible flow problem is relatively straight forward in the sense that it is an extension of the algorithms for solution of generic transport equation. Governing equations for incompressible flows are also similar in form to the generic transport equation. However, there is a small problem due to non-existence of a separate equation for pressure. In this case, there is no equation of state relating pressure, temperature and density, and continuity equation reduces to a kinematic constraint on velocity field.

ANSYS Fluent (Ansys User Manual, 2013) provides comprehensive modeling capabilities for a wide range of incompressible and compressible, laminar and turbulent fluid flow problems. Steady-state or transient analyses can be performed. In ANSYS Fluent, a broad range of mathematical models for transport phenomena (like heat transfer and chemical reactions) is combined with the ability to model complex geometries. Examples of ANSYS Fluent applications include laminar non- Newtonian flows in process equipment; conjugate warm interchanger in turbo machinery and automotive engine components; pulverized coal

combustion in utility boilers; external aerodynamics; flow through compressors, pumps, and fans; and multiphase flows in bubble columns and fluidized beds.

The fundamental governing equations for flow and warm interchanger are the continuity, momentum (Navier-Stokes) and energy equations along with the equations for modeling the turbulence quantities. Based on the mass flow rate and tube dimensions, the calculated Reynolds numbers are found to fall well below the critical value enabling to treat flow as laminar in the present computations of single tube absorber. The conservation equations for laminar flow in an inertial (non-accelerating) reference frame are presented here. The equation for conservation of mass, or continuity equation, can be written as follows:

$$\partial\rho/\partial t + \nabla.(\rho\vec{v}) = S_m \quad (1)$$

The above equation is the general form of the mass conservation equation and is valid for incompressible as well as compressible fl . The source is the mass added to the continuous phase from the dispersed second phase (for example, due to vaporization of liquid droplets) and any user-defined sources. Conservation of momentum in an inertial (non-accelerating) reference frame is described by

$$\partial/\partial t\,(\underline{\rho}\vec{v}) + \nabla.(\underline{\rho}\vec{v}\vec{v}) = -\nabla\underline{p} + \nabla.(\bar{\bar{\tau}}) + \underline{\rho}\vec{g} + \vec{F} \quad (2)$$

where p is the static pressure, is the stress tensor, and and are the gravitational body force and external body forces (e.g., that arise from interaction with the dispersed phase), respectively. $\vec{F}$ also contains other model-dependent source terms such as porous-media and user-defined sources. In the present computational study, only gravitational body force is included. The stress tensor is given by,

$$\bar{\bar{\tau}} = \mu\left[(\nabla\vec{v} + \nabla\vec{v}^T) - \frac{2}{3}\nabla.\vec{v}I\right] \quad (3)$$

Where is the molecular viscosity, I is the unit tensor, and the second term on the right hand side is the effect of volume dilation.

b. ANSYS

ANSYS Fluent solves the energy equation in the following form:

$$\partial/\partial t\,(\rho E) + \nabla.\left(\vec{v}(\rho E + p)\right) = \nabla.\left(k_{eff}\nabla T - \sum_j h_j\overrightarrow{J_j} + \left(\bar{\bar{\tau}}_{eff}.\vec{v}\right)\right) + S_h \quad (4)$$

where keff is the effective conductivity (k+kt, where kt is the turbulent thermal conductivity, defined according to the turbulence model being used), h is the sensible enthalpy for ideal gases and $\vec{J}_j$ is the diffusion flux of species j. The first three terms on the right-hand side of the equation represent energy transfer due to conduction, species diffusion, and viscous dissipation, respectively. Sh includes the heat of chemical reaction, and any other volumetric heat sources you have defined.

$$E = h - \frac{p}{} + \frac{v^2}{2} \quad (5)$$

where sensible enthalpy h is defined for ideal gases as

$$h = \sum Y_j h_j \quad (6)$$

and for incompressible flows as

$$h = \sum_j Y_j h_j + \frac{p}{\bar{\rho}} \quad (7)$$

Yj is the mass fraction of species j and

$$h_j = \int_{T_{ref}}^{T} c_{pj} dT \quad (8)$$

The value used for Tref in the sensible enthalpy calculation depends on the solver and models in use. For the pressure-based solver Tref is 298.15 K except for PDF models in which case Tref is a user input for the species. For the density-based solver Tref is 0 K except when modeling species transport with reactions in which case Tref is a user input for the species. More details of the governing equations can be found in Ansys Fluent 15.0 users guide (2013) and hence not reproduced here for conciseness.

Turbulent flows are encountered in wide range of natural and engineering systems. In fact, flow through all types of fluid machines (whether hydraulic or gas turbines, pumps, compressors or blowers) is invariably turbulent. Main features of turbulent flows are:

Turbulent flows are highly unsteady, three-dimensional and diffusive.

These contain great deal of vorticity and entail considerable dissipation of kinetic energy of the flow.

Turbulent flows fluctuate on a broad range of time and length scales which makes direct numerical simulation of turbulent flows very difficult.

Due to ubiquitous nature and importance in engineering applications, turbulent flows have received considerable attention from theoreticians, experimentalist and numerical analysts over past century. Difficult nature of the problem still fascinates scientists and engineers alike. In numerical simulation, three approaches have emerged for prediction of turbulent flows:

Reynolds Averaged Navier-Stokes (RANS) simulation

Large Eddy Simulation (LES) and

Direct Numerical Simulation (DNS)

c. Solver Settings

RANS simulations are based on time averaging of Navier-Stokes equations. New terms appear in governing equations which are modelled to ensure closure. The modeling reduces the requirements of very fine grids. RANS simulations are work horse of industrial CFD for design analysis.

Their accuracy is dependent on the underlying turbulence models. Numerical discretization techniques for Navier-Stokes equations discussed earlier are equally applicable to turbulent flow simulations (taken as it is, those approaches represent DNS). However, the user need to solve additional algebraic/differential equations in case of RANS and LES which require modeling of additional unknowns Resulting From Averaging (RANS) or filtering (LES).

There are many options available within the solver in providing the turbulence closure using the RANS approach: (i) zero equation (algebraic) models, (ii) half-equation models, (iii) one equation models, (iv) two-equation models, (v) second order models (Reynolds Stress Model) and (vi) algebraic stress models. Among the many turbulence models available, the widely used two equation models are k-- turbulence kinetic energy (k) and turbulence length scale or its equivalents. These eddy-viscosity methods are based on similarity reasoning that turbulence is a physical concept connected to the viscosity. For the present study, a suitable turbulence model was identified, from open literature. Several turbulence models were tested by numerous researchers for the end- wall film cooling problem and reported in the open literature. The laminar k- turbulence model was used for the present simulations on the active system flow configuration used in the forced convection setup.

Steady state, three dimensional, incompressible heat and mass transfer simulations using the RANS (Reynolds Averaged Navier Stokes) approach with Realizable k- equations as the turbulence closure will be carried out. The results from the computations will be validated with the available experimental results and further results from the numerical predictions will be presented. The boundary conditions used for the present CFD simulations are shown in Figure 1.

The boundary conditions used can be listed as below:

Constant heat flux, q" = Specified W/m2: Radiator tube top and bottom surface Constant heat flux, q" = 0 W/m2 : Radiator tube side surfaces Specified mass flow rate, = c kg/s : Heat exchanger inlet Specified static pressure, Ps = c Pa. : Heat exchanger outlet

The convergence criteria for the computational solution were determined based on scaled residuals for the equations of continuity, momentum, energy equations and turbulence quantities. The scaled residuals for solution convergence were set to 10e-5 for all governing equations and turbulence quantities. The solution was considered to be converged when all

the scaled residuals were less than or equal to the prescribed value. Computations were carried out until the steady state was reached.

For a few cases, the convergence was not achieved to the desired accuracy specified. In those cases, the iteration was continued further to a stage such that the results did not vary even after 500 iterations, thus achieving iterative convergence.

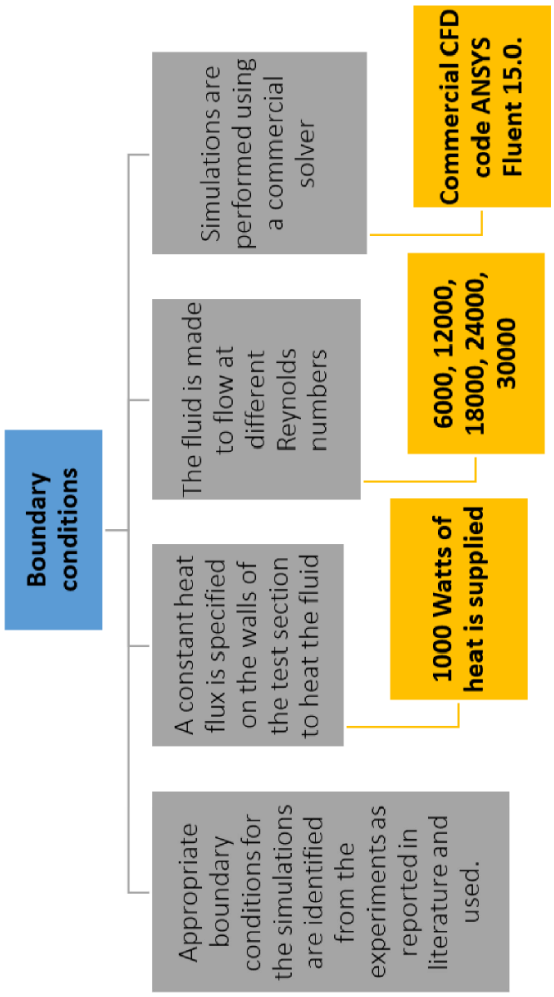

Figure 1 Boundary conditions used

7. Conclusion

This Research paper discusses the several computational investigations carried out for the present experimental domain and the results are compared. Suitable solver setting, turbulence model was identified from the literature and the boundary conditions for the simulations were taken from the present experiment. Numerical investigations were based on 3D steady state conjugate warm interchangeranalysis to infer flow and warm interchangercharacteristics of the absorber fluid using the FVM based code Ansys Fluent ® 15.0. The objective of the CFD analysis was to validate the computational methodology adopted

References

[1] D. Littlefield, P. Desai, Buoyant laminar convection in a vertical cylindrical annulus, Trans. ASME 108 (2004).

[2] V. Bansal, R. Misra, G.D. Agrawal, J. Mathur, Performance analysis of earth-epipeeair heat exchanger for summer cooling, Energy and Buildings 42 (2010) 645e648.

[3] R. Kumar, S. Ramesh, S.C. Kaushik, Performance evaluation and energy conservation potential of earth- air-pipe system coupled with non-air- conditioned building. Building Environ. 38:807e813.

[4] M.K. Ghosal, G.N. Tiwari, D.K. Das, K.P. Pandey, Modeling and compar- ative thermal performance of ground air collector and earth air heatexchanger for heating of greenhouse, Energy and Buildings 37 (2005) 613e621.

[5] F.A. Ajmi, D.L. Loveday, V. Hanby, The cooling potential of eartheair heat exchangers for domestic buildings in a desert climate, Building Environ. 41 (2006) 235e244.

[6] Y. Wu, G. Gan, A. Verhoef, P.L. Vidale, R.G. Gonzalez, Experimental measure- ment and numerical simulation of horizontal-coupled slinky ground source heat exchangers, Appl. Therm Eng. 30 (2010) 2574e2583.

[7] S. Florides, S. Kalogirou, Ground heat exchangers: a review of systems, models and applications, Renewable Energy 32 (2007) 2461e2478.