NOVATEUR PUBLICATIONS
INTERNATIONAL JOURNAL OF INNOVATIONS IN ENGINEERING RESEARCH AND TECHNOLOGY
[IJIERT]
ISSN: 2394-3696
VOLUME 3, ISSUE 1, JAN.-2016

CFD ANALYSIS OF HEAT TRANSFER IN HELICAL COIL TUBE IN TUBE HEAT EXCHANGER

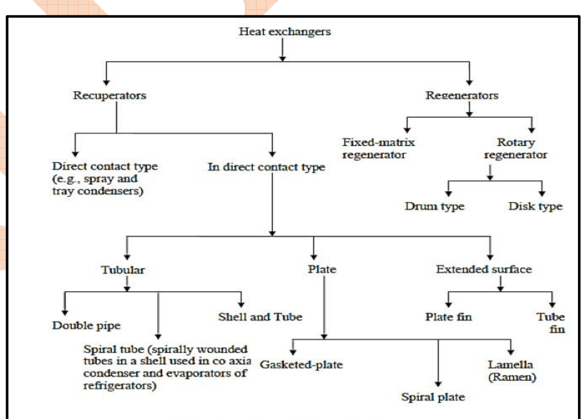
Pramod Deshmukh,
Mechanical Dept. of BVCOE, India
Vikram D Patil,
Mechanical Dept. of BVCOE, India
Prof. Baviskar Devakant,
Mechanical Department of BVCOE, India

ABSTRACT

Heat Exchanger is generally used for transfer the heat from one end to another end. There is various type of heat exchanger in that helical coil tube type heat exchanger shows better result among them. In helical coil tube heat exchanger two types of flow generally used Parallel flow and counter flow. In this paper by using counter flow analysis of helical coil tube is done. The contours of temperature of the tube were calculated and plotted using ANSYS FLUENT 15.0. Copper was chosen as the as metal for the construction of the helical tube. The fluid flowing through the tube was taken as water. First law of thermodynamics concept is play vital role in helical coil tube heat exchanger**.**

INTRODUCTION

The most important fluid flow heat exchangers are HVAC, process industry, refrigeration etc. The purpose of constructing a heat exchanger is to get an efficient method of heat transfer from one fluid to another, by direct contact or by indirect contact. There are three mode of heat transfer 1.Conduction 2.Convection 3.Radiation.Heat transfer is negligible in radiation as compare to conduction and convection. Conduction takes place when the heat from the high temperature fluid flows through the surrounding solid wall. The conductive heat transfer can be maximised by selecting a minimum thickness of wall of a highly conductive material. But convection is plays the major role in the performance of a heat exchanger. Forced convection in a heat exchanger transfers the heat from one moving stream to another stream through the wall of the pipe. The cooler fluid removes heat from the hotter fluid as it flows along or across it



Classification of Heat Exchanger

Figure 1 Classification of heat Exchanger

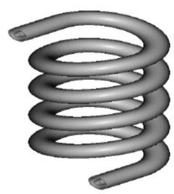
TUBULAR HEAT EXCHANGERS

Tubular heat exchangers are built of mainly of circular tubes there are some other geometry has also been used in different applications. This design can be modified by length, diameter and physical arrangement. This type is used for liquid-to-liquid (phase changing like condensing or evaporation) heat transfer. Again this type is classified into shell and tube, double pipe and spiral tube heat exchangers.

ISSN: 2394-3696
VOLUME 3, ISSUE 1, JAN.-2016

DOUBLE PIPE HEAT EXCHANGER

The double pipe or the tube in tube type heat exchanger consists of one pipe placed concentrically inside another pipe having a greater diameter. The flow in this configuration cane be of two types: parallel flow and counter-flow. It can be arranged in a lot of series and parallel configurations to meet the different heat transfer requirements. Double coil heat exchanger is widely used; knowledge about the heat transfer coefficient, pressure drop, and different flow patterns has been of much importance. The curvature in the tubes creates a secondary flow, which is normal to the primary axial direction of flow. This secondary flow increases the heat transfer between the wall and the flowing fluid. And they offer a greater heat transfer area within a small space, with greater heat transfer coefficients. The two basic boundary conditions that are faced in the applications are constant temperature and the constant heat flux of the wall.



Double pipe helical coil Close-up of double pipe
Fig 2 Fig 3

HEAT TRANSFER COEFFICIENT

The value of 'h' depends upon the properties of fluid Convective heat transfer, is the transfer of heat from one place to another by the movement of fluids due to the difference in density across a film of the surrounding fluid over the hot surface. Through this film heat transfer takes place by thermal conduction and as thermal conductivity of most fluids is low, the main resistance lies there. Heat transfer through the film can be enhanced by increasing the velocity of the fluid flowing over the surface which results in reduction in thickness of film. The equation for rate of heat transfer by convection under steady state is given by,

$$Q = h\,A\,(T_w - T_{atm}) \;\ldots\ldots\ldots\ldots\; (1)$$

Where

H-is the film coefficient or surface coefficient (W/m2.K).

A is the area of the wall

Tw is the wall temperature

T atm is surrounding temperature.

Within the film region; hence it is called 'Heat Transfer Coefficient'. It depends on the different properties of fluid, dimensions of the surface and velocity of the fluid flow (i.e. nature of flow).

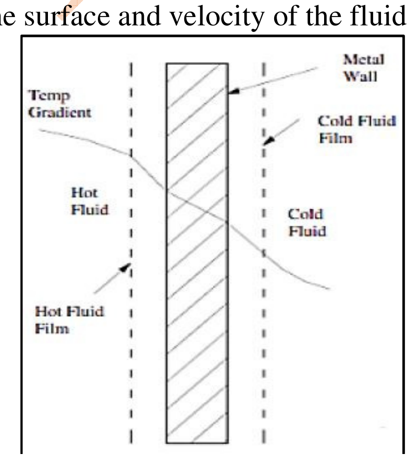


Fig 4 Wall Convection

AIM OF THE PRESENT WORK

The design of a helical coil tube in tube heat exchanger has been facing problems because of the lack of experimental data available regarding the behaviour of the fluid in helical coils and also in case of heat transfer data, which is not the case in Shell & Tube Heat Exchanger. For that we are doing numerical analysis was carried out to determine the heat transfer characteristics for a double-pipe helical heat exchanger by varying the different parameters like different temperatures and diameters of pipe and coil and also to determine the fluid flow pattern in helical coiled heat exchanger. The objective of the project is to obtain a better and more quantitative insight into the heat transfer process that occurs when a fluid flows in a helically coiled tube. The study also covered the different types of fluid flow range extending from laminar flow through transition to turbulent flow. The materials for the study were decided and fluid taken was water and the material for the pipe was taken to be copper for its better conducting properties.

DESIGN METHODOLOGY

COMPUTATIONAL FLUID DYNAMICS: -

Usually abbreviated as CFD, is a branch of fluid mechanics that uses numerical analyse and algorithms to solve and analyse problems that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions with high-speed supercomputers, better solutions can be achieved. Ongoing research yields software that improves the accuracy and speed of complex simulation scenarios such as transonic or turbulent flows. Initial experimental validation of such software is performed using a wind tunnel with the final validation coming in full-scale testing, e.g. flight tests. The fundamental basis of almost all CFD problems are the Navier–Stokes equations which define any single-phase (gas or liquid, but not both) fluid flow. These equations can be simplified by removing terms describing viscous actions to yield the Euler equations further simplification, by removing terms describing vortices yields the full potential equations. Finally, for small perturbations in subsonic and supersonic flows (not transonic or hypersonic) these equations can be linearized to yield the linearized potential equations Computational fluid dynamics (CFD) study of the system starts with the construction of desired geometry and mesh for modelling the dominion. Generally, geometry is simplified for the CFD studies. Meshing is the discretization of the domain into small volumes where the equations are solved by the help of iterative methods. Modelling starts with the describing of the boundary and initial conditions for the dominion and leads to modelling of the entire system. Finally, it is followed by the analysis of the results, discussions and conclusions.

GEOMETRY

Heat exchanger is built in the ANSYS workbench design module. It is a counter-flow heat exchanger. First, the fluid flow (fluent) module from the workbench is selected. The design modeller opens as a new window as the geometry is double clicked.

SKETCHING

Out of 3 planes, XY-plane, YZ-plane and ZX-plane, the YZ-plane is selected for the first sketch. A 4 inch line for the height of the helical structure is made. A new plane is created in reference with the YZ-plane which is termed as plane 4. 4 new sketchers are added under the new plane, i.e. plane 4. In sketch 2, a circle of diameter 0.545 inch at a distance of 3 inch from origin. In sketch 3, two circles of diameters 0.545 inch and 0.625 inch are made concentric to previous circle. In sketch 4, two circles of diameters 0.625 inch and 0.785 inch are made concentric to previous circles. In sketch 5, two circles of diameters 0.785 inch and 0.875 inch are made concentric to previous circles.

SWEEP

Sketch 2, 3, 4 & 5 are swept along the line made in sketch made in sketch 1 using the” add frozen” operation to construct the 3D model with different parts. The helical sweep is of 2 turns because the twist specification is defined in number of turns.

MERGING

After sweep operation, it will show the model as 4 parts and 4 bodies. For merge operation, all the 4 parts are selected using control and merged as 1 part. At the end it will show as 1 part and 4 bodies. The 4 bodies within 1 part are named as follows:-

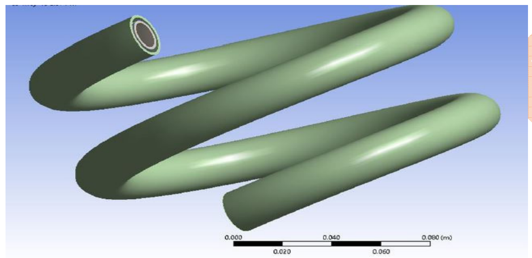


Fig 5 Original Geometry

ANALYSIS OF CFD

MESH:-Initially a relatively coarser mesh is generated. This mesh contains mixed cells (Tetra and Hexahedral cells) having both triangular and quadrilateral faces at the boundaries. Care is taken to use structured hexahedral cells as much as possible. It is meant to reduce numerical diffusion as much as possible by structuring the mesh in a well manner, particularly near the wall region. Later on, a fine mesh is generated. For this fine mesh, the edges and regions of high temperature and pressure gradients are finely meshed.

Y+ VALUES:-y + values play a significant role in turbulence modelling for the near wall treatment. y+ is a non-dimensional distance. It is frequently used to describe how coarse or fine a mesh is for a particular flow pattern. It determines the proper size of the cells near domain walls. The turbulence model wall laws have limitations on the y+ value at the wall. For instance, the standard K-epsilon model requires a wall y+ value between approximately 300 and 100. A faster flow near the wall will produce higher values of y+, so the grid size near the wall must be decreased. y+ values for different wall treatments are given in

Wall treatment method	Recommended y^+ values	Used y+ values at tube walls
Standard wall functions	$30 < y^+ < 400$	$y^+ < 5$
Non-equilibrium wall functions	$30 < y^+ < 100$	$y^+ < 5$
Low Reynolds number model	$y^+ \cong 1$	$y^+ < 1$

Table 1 y+ Values for Different Wall Treatments

The mesh details view gave us the following information:
Relevance centre: fine meshing
Smoothing: high
Size: 4.033e-005m to 8.066e-005m
Pinch tolerance: 3.6297e-005m
Nodes: 586300
Elements: 53170

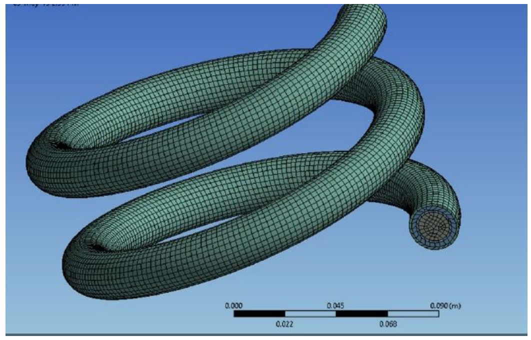


Fig 7 Mesh

NAMED SELECTION:-The different surfaces of the solid are named as per required inlets and outlets for inner and outer fluids. The outer wall is named as insulation surface.

Part number	Part Of The Model	State Type
1	Inner_Fluid	Fluid
2	Inner_Pipe	Solid
3	Outer_Fluid	Fluid
4	Outer_Pipe	Solid

Table 2 Name Selection

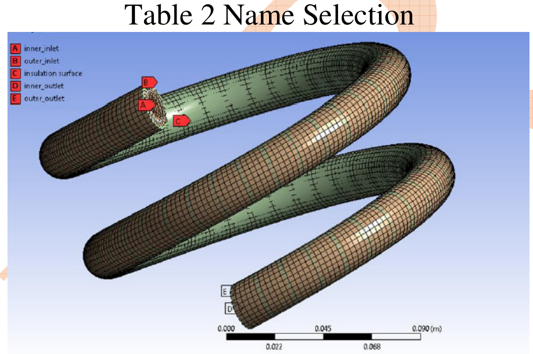


Fig 8 Name Selection

SOLUTION:-

PROBLEM SETUP:-The mesh is checked and quality is obtained. The analysis type is changed to Pressure Based type. The velocity formulation is changed to absolute and time to steady state. Gravity is defined as y = -9.81 m/s2

MODELS:-Energy is set to ON position. Viscous model is selected as “k-ε model (2 equation). Radiation model is changed to Discrete Ordinates.

MATERIALS:-The create/edit option is clicked to add water-liquid and copper to the list of fluid and solid respectively from the fluent database.

CELL ZONE CONDITIONS:-The parts are assigned as water and copper as per fluid/solid parts.

BOUNDARY CONDITIONS:-Boundary conditions are used according to the need of the model. The inlet and outlet conditions are defined as velocity inlet and pressure outlet. As this is a counter-flow with two tubes so there are two inlets and two outlets. The walls are separately specified with respective boundary conditions. No slip condition is considered for each wall. Except the tube walls each wall is set to zero heat flux condition. The details about all boundary conditions can be seen in the table 3 as given below.

-	Boundary Condition Type	Velocity Magnitude	Turbulent Kinetic Energy	Turbulent Dissipation Rate	Temperature
Inner Inlet	Velocity Inlet	0.9942 m/s	0.01 m^2/s^2	0.1 m^2/s^3	348 K
Inner Outlet	Pressure Outlet	-	-	-	-
Outer Inlet	Velocity Inlet	0.8842 m/s	0.01 m^2/s^2	0.1 m^2/s^3	283 K
Outer Outlet	Pressure Outlet	-	-	-	-

Table 3 Boundary Conditions Table

REFERENCE VALUES:-The inner inlet is selected from the drop down list of “compute from”. The values are:

Area = 1 m2
Density = 998.2 kg/m3
Length = 39.37008 inch
Temperature = 348 K
Velocity = 0.9942 m/s
Viscosity = 0.001003 kg/m-s
Ratio of specific heats = 1.4

SOLUTION METHODS:-

Scheme = Simple
Gradient = Least Square Cell Based
Pressure = Standard
Momentum = Second Order Upwind
Turbulent Kinetic Energy = Second Order Upwind
Turbulent Dissipation Rate = Second Order Upwind

SOLUTION CONTROL AND INITIALIZATION:-

Under relaxation factors the parameters are

Pressure = 0.3 Pascal
Density = 1 kg/m3
Body forces = 1 kg/m2s2
Momentum = 0.7 kg-m/s
Turbulent kinetic energy = 0.8 m2/s2

Then the solution initialization method is set to Standard Initialization whereas the reference frame is set to Relative cell zone. The inner inlet is selected from the compute from drop down list and the solution is initialized.

III.4.9Measure of Convergence:- It is tried to have a nice convergence throughout the simulation and hence criteria is made strict so as to get an accurate result. For this reason residuals are given as per the table 4 that follows.

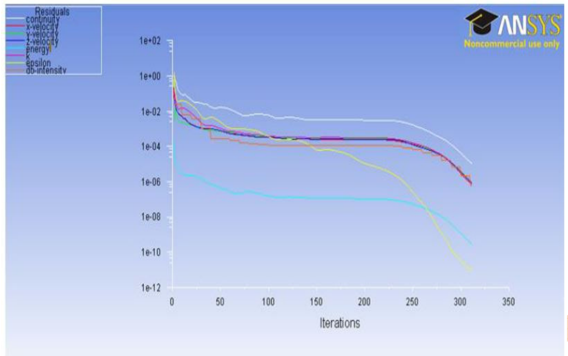


Fig 9Residuals Scaled

Variable	Residual
x-velocity	10^{-6}
y-velocity	10^{-6}
z-velocity	10^{-6}
Continuity	10^{-6}
Specific dissipation energy/ dissipation energy	10^{-5}
Turbulent kinetic energy	10^{-5}
Energy	10^{-9}

Table 4 Residuals

CONTOURS:-The temperature, pressure and velocity distribution along the heat exchanger can be seen through the contours.

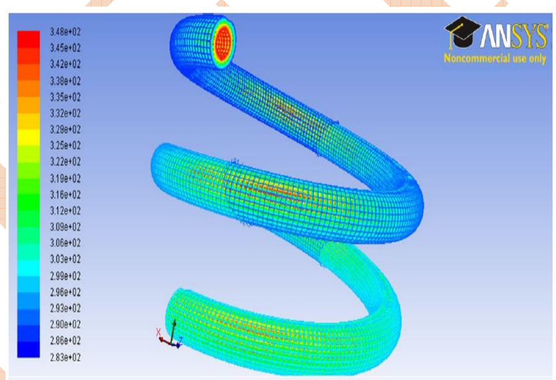


Fig 10Contours of Static Temperature

CONCLUSIONS

A CFD package (ANSYS FLUENT 15.0) was used for the numerical study of heat transfer characteristics of a helical coiled double pipe heat exchanger of the counter-flow. The CFD results when compared with the experimental results from different studies and were well within the error limits. The simulation was carried out for water to water heat transfer characteristics and different inlet temperatures were studied. Characteristics of the fluid flow were also studied for the constant temperature and constant wall heat flux conditions.

REFERENCES

[1]. B. Chinna Ankanna, B. Sidda Reddy (2014),” Performance Analysis of Fabricated Helical Coil Heat Exchanger” International Journal of Engineering Research, Volume No.3 Issue No: Special 1, pp: 33-39.

[2]. Pramod S. Purandare, Mandar M. Lele, Rajkumar Gupta (2012)“Parametric Analysis of

Helical Coil Heat Exchanger" International Journal of Engineering Research & Technology (IJERT) Vol. 1 Issue 8.
[3]. Kapil Dev, Kuldeep Singh Pal, Suhail A. Siddiqui (2014) "An Empirical Study of Helical Coil Heat Exchanger Used in Liquid Evaporization and Droplet Disengagement for a Laminar Fluid Flow" INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH TECHNOLOGY.
[4]. Shiva Kumara, K.Vasudev Karanth (2013) "Numerical Analysis of a Helical Coiled Heat Exchanger using CFD" International Journal of Thermal Technologies Vol.3, No.4 (Dec 2013).
[5]. Pablo Coronel K.P. Sandeep (2008) "Heat Transfer Coefficient in Helical Heat Exchangers under Turbulent Flow Conditions" International Journal of Food Engineering Volume 4, Issue 1.