

# International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Website: [www.ijirset.com](http://www.ijirset.com)

Vol. 6, Issue 4, April 2017

## CFD Analysis of Heat Transfer on Duct

Deepak Kumar Gupta

M. E. Scholar, Department of Mechanical Engineering, RITEE, Raipur, India

**ABSTRACT:** To analyze the characteristics of triangular duct plate fin array, a rectangular duct is considered on which a triangular duct plate fins are inserted for analysis. A 3-D model has been developed and SST  $k-\omega$  turbulent model is to be selected for simulation because it gives better turbulent model. The Results shows a good relation between Outlet temperature and Inlet temperature for enhancing the heat transfer. From analysis, triangular duct plate fin array inserts increase the heat transfer rate with expectation of mass flow rate of fluid.

**KEYWORDS:** Rectangular Duct, triangular duct plate fins array, CFD analysis, modeling.

### I. INTRODUCTION

Computational Fluid Dynamics (CFD) is the computer-based simulation to analyze systems involving fluid flow, heat transfer etc. In CFD procedure firstly a numerical model is constructed based on mathematical equations, which describe the flow

Heat exchanger is process equipment designed for the effective transfer of heat energy between two fluids a hot fluid and a coolant. The purpose may be either to remove heat from a fluid or to add heat to a fluid. Heat exchangers used in boilers (evaporators), super-heaters and condensers, Automobile radiators, oil cooler of heat engine, water and air heater or coolers, cryogenic and various applications.

The heat exchangers used in liquefiers have the high effectiveness; the minimum effectiveness of heat exchanger devices required in regenerative refrigerators. In aircrafts the demand on volume and weight of the heat exchanger must be as soon as minimum but the Performance may be less. These requirements have led to the development of other unique heat exchangers known as compact heat exchangers.

Many developments in the field of CFD along with its ability to model complex phenomena and growing popularity of such software's have widened the range of application of CFD. It is applied in a wide range of industries including design analysis, mechanical process, and power, metallurgical, biomedical.

CFD has also been successfully used in modeling various multiphase flow systems, particularly gas-solid mixtures, although some limitations still exist.

### II. RELATED WORK

C. Mageswaran, R. Muthukumaran, R. Karthikeyan, R. Rathnasamy, They have investigated heat transfer of pin-fin heat sinks with circular, square and diamond cross sections. Ensemble of pin-fin heat sinks with in-line and staggered arrangements were designed and tested experimental. They examined the fin density effect on heat transfer and friction characteristics and concluded that in all, the circular pin-fin arrangement shows an appreciable influence of fin density and no effect is seemed in case of square fin geometry.

Anchit Gupta, Sushant Samir, They have investigated the Nusselt number and flow characteristics in the 2-D rectangular channel with built-in triango-circular inserts numerically in the periodically fully developed regime using ANSYS 15.0. The calculations were performed for a Reynolds number 50 and 100 angle of tip triango-circular inserts  $60^\circ$ . The authors concluded that by using triango-circular insert the heat transfer rate is significant increase. There was 6.025% increase in nusselt number at Reynolds number 50 and 12.18% increase in nusselt number at Reynolds number 100. There was 12.18% % increase in the average Nusselt number by using triango-circular insert with an angle of  $60^\circ$  and  $Re = 100$ .

# International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Website: [www.ijirset.com](http://www.ijirset.com)

Vol. 6, Issue 4, April 2017

Ruoxu Jia, Junling Hu, Xingguo Xiong, They have simulated a counter flow plate fin parallel heat exchanger. A repeating unit cell of the multi channel heat exchanger has taken as the computational domain, on which a cold channel and a hot channel were separated by plates. The model was simulated in COMSOL, Hot oil and cold water was separated by two parallel channels in opposite directions and analyzed the distributions of temperature, velocity, and pressure, and concludes that they are also used for finding the performance of the heat exchanger. A parametric study over channel length was also applied to seem the effect on heat transfer effectiveness and pressure drop.

Ertan Buyruk, Koray Karabulut, They have numerically studied the rectangular fins with a 30° and 60° angle and 10 mm offset from the horizontal direction perpendicular to flow for heat transfer enhancement in a plate fin heat exchanger, The rectangular fins were mounted on a flat plate channel. Numerical computations have performed in Fluent software program to solving a steady, three-dimensional Navier-Stokes equation and an energy equation. The study was carried out at Re=400 inlet temperatures, velocities of cold and hot air are fixed at 300 K, 600 K and 1.338 m/s, 0.69 m/s, respectively. The heat transfer enhancement with fins of 30° and 60° for different values of the Reynolds number with 300 and 800 and for varying fin heights, fin intervals and temperature distributions on the hot and cold fluid sides of the channel outside surface have also investigated in the parallel and counter flow arrangement. Their authors conclude that heat transfer increased by about 9 % at the exit of the channel with a fin angle of 60° compared to the channel without fins for parallel flow.

Masoud Asadi, Arash Nadali, They have studied pressure drop and heat transfer coefficient for Finned-tube heat exchangers, plate-fin heat exchanger with Louvered fins and Offset strip fins, and suggested that the Colburn and friction factors can be used to predict the convective coefficient and pressure drop. They applied optimization to minimize pressure drops and simultaneously improve heat transfer characteristics. They author also conclude that friction and colburn factors decrease with increasing Reynolds number, and Reynolds number of 1000 to 2000 there was a transition region, and after this Reynolds number functions have decreased moderately. Reynolds number, friction factor is ten times more than Colburn factor.

### III. PROPOSED MODEL AND ANALYSIS

CFD analysis of heat exchanger belongs to modeling of rectangular duct and triangular duct plate fin array. It provides the detailed information about the mathematical model of rectangular duct with triangular duct plate fin array inserts. Computational Fluid Dynamics (CFD) is the computer-based simulation to analyze systems involving fluid flow, heat transfer etc. In CFD procedure firstly a numerical model is constructed based on mathematical equations, which describe the flow. These equations are then solved using a computer programmed in order to obtain the flow variables throughout the flow domain. Since the advent of the digital computer, CFD has received extensive attention and widely used to study various aspects of fluid dynamics.

#### Mathematical Model

The available finite volume procedures for parallel flows and boundary layer are employed to solve the governing partial differential equations. Some simplifying assumptions are required for applying the conventional flow equations and energy equations to model the heat transfer process in rectangular duct with insert of triangular duct plate fin arrays. For turbulent, steady, and incompressible flow with constant properties, while neglecting the natural convection and radiation, the three-dimensional equations of continuity, momentum, and energy, in the fluid region are solved. These equations are as below:

Continuity Equation

$$\frac{\partial(\rho\vartheta)}{\partial t} + \vartheta\nabla \cdot (\rho\vartheta) = -\nabla P + \nabla \cdot \tau + F + \rho g \quad (3.1)$$

Momentum equation

$$\frac{\partial(\rho E)}{\partial t} \nabla \cdot (\vartheta(\rho E + p)) = \nabla \cdot (k_{eff}\nabla T + \tau_{eff} \cdot \vartheta) \quad (3.2)$$

Energy Equation

$$-\rho u'_i u'_j = \mu t \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \left( \rho k + \mu t \frac{\partial u_k}{\partial x_k} \right) \delta_i \quad (3.5)$$

The Second Order Upwind Scheme was used to discretization the convective term. The linkage between the velocity and pressure was computed using the SIMPLE algorithm.

# International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Website: [www.ijirset.com](http://www.ijirset.com)

Vol. 6, Issue 4, April 2017

## CFD PROCEDURE

All commercial CFD packages involve sophisticated user interfaces to input parameters and to examine the results. Hence all the codes consist of three main elements:

1. Pre-processor
2. Solver
3. Post-processor

Pre-processing is the input of a flow problem to a CFD program by means of an operator friendly interface and the subsequent transformation of this input into a form suitable for use by the solver. The steps involved in this are:

- a) Definition of geometry.
- b) Grid generation.
- c) Selection of the phenomena or system to be modeled.
- d) Definition of fluid properties.
- e) Boundary conditions specification.

Solver involves the following steps:

- a) Approximation of the unknown flow variables by means of simple functions.
- b) Discretization by substitution of the approximations into the governing flow equations.
- c) Solution of algebraic equations.

Post-processor- All the leading CFD packages are equipped with versatile data visualization tools. These include:-

- a) Domain geometry and grid display
- b) Vector plots
- c) Line and shaded contour plots
- d) Color postscript output

The mathematical modeling of a flow problem is achieved basically through three steps:

- a) Developing the governing equations describing the flow;
- b) Discretization of the governing equations; and
- c) Solving the resulting numerical equations.

### Governing equations:

1. Unsteady state 3-D equation of continuity: - (Equation 3.2) where  $\rho$  is the fluid density,  $t$  is time, and  $u$  is velocity.
2. Momentum equation: - (Equation 3.3) where  $P$  is pressure,  $\mu$  is fluid viscosity,  $x$  is the coordinate and the subscript  $i$  indicates the Cartesian coordinates.
3. Energy equation:- (Equation 3.4) where  $k$  is the thermal conductivity,  $T$  is the temperature, is the dissipation term

And  $S$  is the source term.

The governing equations shown earlier are partial differential equations (PDEs). The PDEs must be transformed into numerical equations containing only numbers and no derivatives. This process of producing a numerical analogue to the PDEs is called 'numerical discretization'. Various techniques used for discretization are the finite difference method, the finite element method, and the finite volume method.

The commercial CFD software used for the current project is ANSYS 15.0.

The CFD Process is carried out in following steps,

- I. Model generation
- II. Meshing
- III. Setup (boundary condition)
- IV. Solution
- V. Result

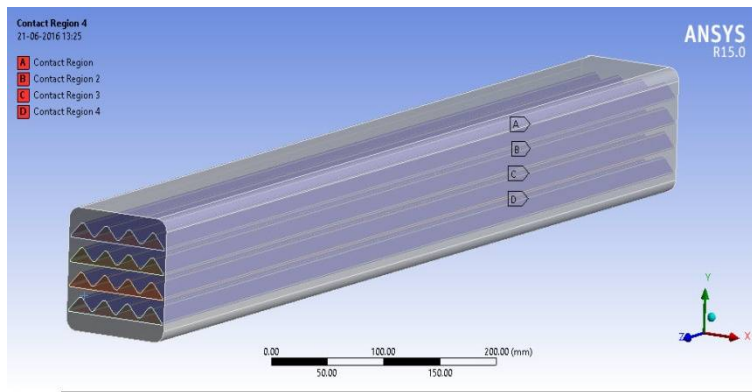


Figure 1: Complete Drawing Model of Rectangular duct and triangular duct plate fins array inserts heat exchanger

**I. Model Generation**

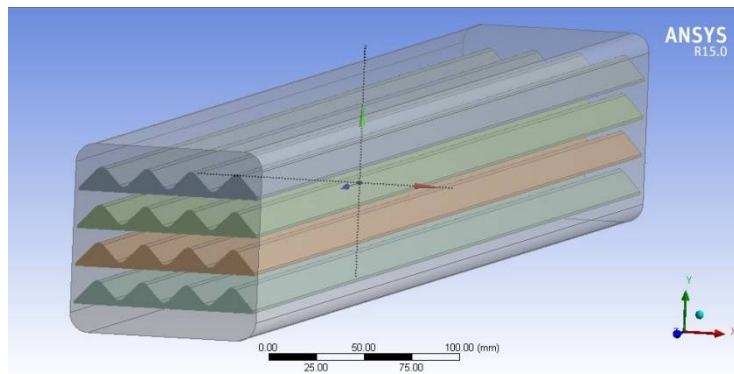


Figure 2: Model of rectangular duct and triangular duct plate fin arrays

**II. Meshing**

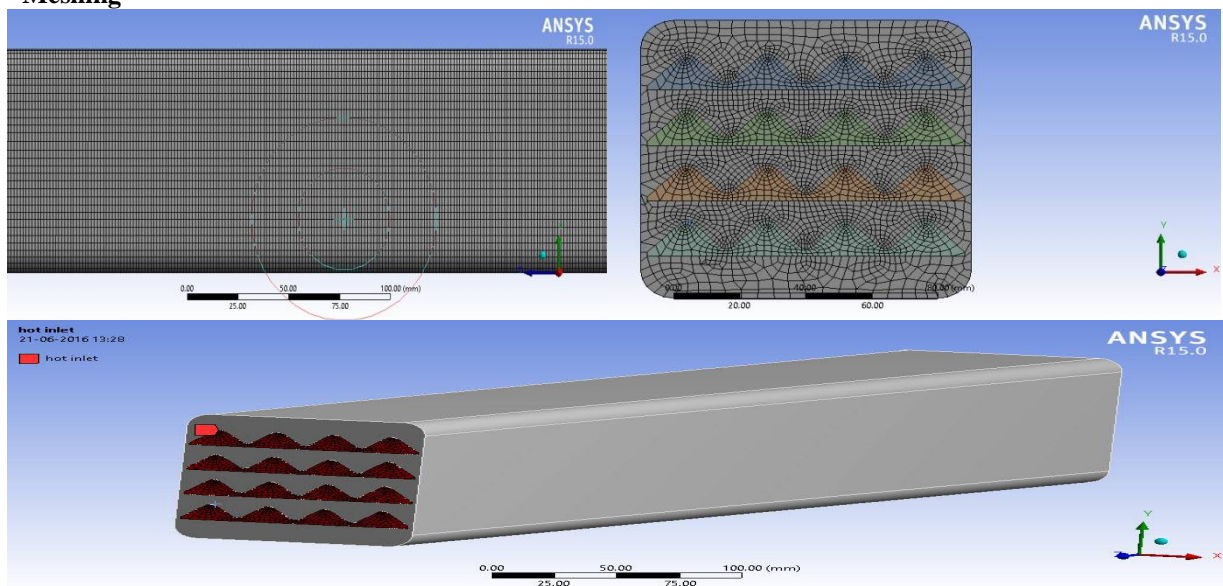


Figure 3: Meshed view of rectangular duct and triangular duct plate fin arrays

## International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Website: [www.ijirset.com](http://www.ijirset.com)

Vol. 6, Issue 4, April 2017

After completing the meshing process of 'Face Zone' and 'Cell Zone' the definition has been completed. In this problem following 'Face Zones' have been chosen, 'velocity-Inlet', velocity-outlet' and 'Wall'. 'Cell-Zone' has been identified as 'Fluid'.

After completion of meshing and zone definition in Gambit, the work has been saved and exported as a mesh file extension to use in CFD.

Now to complete the simulation of this flow problem following parameters have been set in the 'CFD' software,

- A 3-dimensional flow problem.
- Steady flow takes place through the region.
- Semi Implicit Pressure based Solver chosen.
- Absolute velocity considered.
- To define the 'Viscous Model', k-omega (2-eqn) model has been considered with SST.

To complete the analysis of flow through the defined flow region it is necessary to mention the property of fluid. The fluid which has been considered here is Nitrogen gas as per reference. Property of Nitrogen gas has been mentioned in Table 3.3.

### III. Boundary Condition

1. Inlet face was given with specified velocity condition and temperature is 358.83 K.
2. The wall of the test section known as heated wall was given uniform heat flux along its length.
3. The physical property of fluid which is taken gas (Nitrogen) was constant and was evaluated at mean temperature.
4. The flow region was specified as fluid region so that it is solved by heat convection equation.

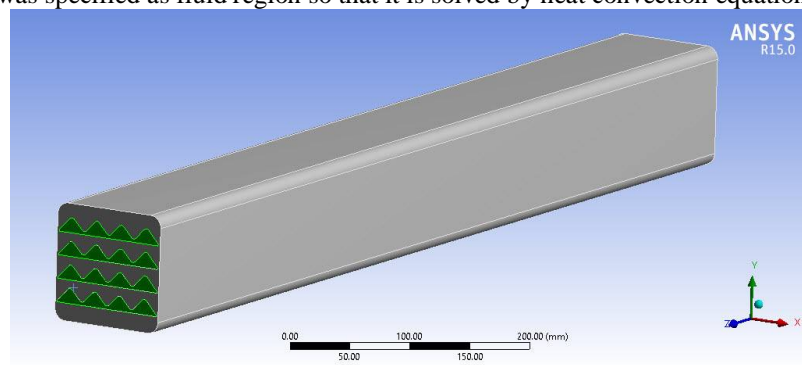


Figure 4: Complete Drawing of Rectangular duct with insert of triangular duct plate fin array heat exchanger with Flow

### IV. Solution

This problem has been solved with 'SIMPLE' algorithm. The full form of 'SIMPLE' is 'Semi Implicit Method for Pressure Linked Equation'. Before we start simulating we should mention the convergence value. Convergence is the difference of value of any quantity evaluated in any two consecutive steps in the course of simulation. It has been kept for all parameters except energy  $10^{-3}$  and energy  $10^{-6}$  for better result. After convergence reached, the software is ready for post-processing.

### V. Post-Processing

In post-processing following quantity would be evaluated or displayed for this problem.

- a) Graph depicting Outlet temperature at different Inlet temperature.
- b) Temperature Distribution along the length.



# International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Website: [www.ijirset.com](http://www.ijirset.com)

Vol. 6, Issue 4, April 2017

## IV. SIMULATION AND RESULT

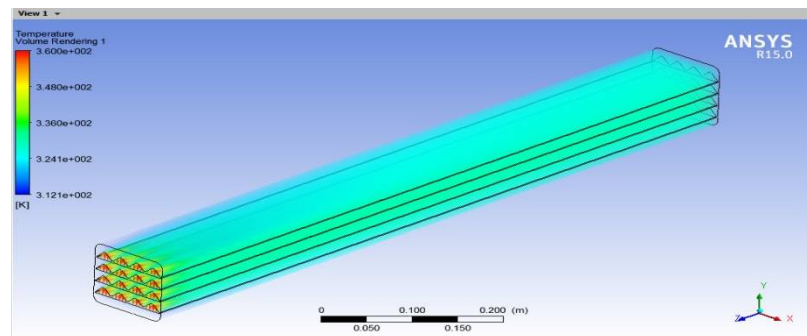
The dimensions for modeling of a rectangular duct heat exchanger are shown in table – 1, and fluid properties and thermal properties also considered as shown on table- 2.

Table - 1 Dimensions of Rectangular Duct

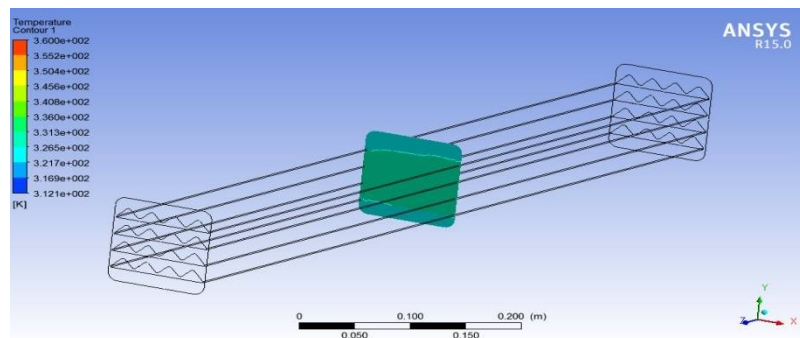
Parameter	Symbols	Units	Dimensions
Core Length	L1	mm	900
Core Width	L2	mm	100
Core Height	L3	mm	100
Plate Thickness	tw	mm	6

Table - 2 Thermal and Fluid Properties of Fluid

Parameter	Symbols	Units	Hot	Cold
Fluid used			Hot fluid	Cold fluid
Specific heat	cp	J/kg- K	5197.5	5197.5
Viscosity	$\mu$	Pa-s	0.0000154	0.0000148
Prandtl number	Pr		0.6685	0.67
Density	$\rho$	kg/m <sup>3</sup>	1.7404	0.2808
Inlet temperature	ti	K	358.83	313.94
Mass flow rate	$\dot{m}$	Kg/s	4.5	5



(a)



(b)

Figure 5: (a) Average outlet temperature at hot water (b) at mid section

## International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Website: [www.ijirset.com](http://www.ijirset.com)

Vol. 6, Issue 4, April 2017

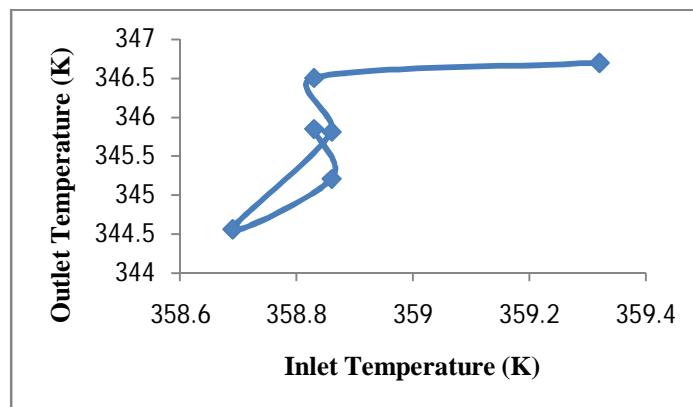


Figure 6: Graph represents CFD Analysis Inlet Temperature Vs Outlet Temperature of Hot fluid

### V. CONCLUSION

From the above graph it is very much clear that insertion of a triangular duct plate fin array in a rectangular duct or pipe increase the thermal performance of the duct or pipe. The cold fluid's temperature at the outlet of the duct or pipe has been increased. The reason for the increment of these parameters is that, due to triangular duct plate fin array a more surface contact is developed in the duct or pipe, which helps the fluid to take more heat from the fluid.

So it may be concluded that, Plate fin heat exchanger having rectangular duct on with triangular duct plate fins array are inserted disturb the flow, due to this the heat transfer area increase.

### REFERENCES

- [1] Mageswaran, C., Muthukumar, R., Karthikeyan, R., and Rathnasamy, R., "HEAT TRANSFER AND FRICTION IN RECTANGULAR DUCT WITH PIN-FIN ARRAY," IJERT, Volume-5, Issue-2, 2016.
- [2] Buyruk, Ertan, Karabulut, Koray, "NUMERICAL INVESTIGATION INTO HEAT TRANSFER FOR THREE-DIMENSIONAL PLATE FIN HEAT EXCHANGERS WITH FINS PLACED PERPENDICULAR TO FLOW," Transactions of famena xxxvii-2, 2013
- [3] Gupta, Anchit, Samir, Sushant, "HEAT TRANSFER ENHANCEMENT IN A LAMINAR CHANNEL FLOW WITH BUILT-IN TRIANGULAR CIRCULAR INSERTS," International Journal of Emerging Research in Management & Technology, Volume-4, Issue-5, 2015.
- [4] Jia, Ruoxu, Hu, Junling, Xiong, Xingguo "MODELING OF A COUNTER FLOW PLATE FIN HEAT EXCHANGER," Excerpt from the Proceedings of the 2014 COMSOL Conference in Boston, Boston, 2014.
- [5] Asadi, Masoud, Nadali, Arash, "STUDY ON THE FUNCTIONS OF FRICTION AND COLBURN FACTORS IN COMPACT HEAT EXCHANGER," Wyo Academic Journal of Engineering & Technology Research, Volume-1(4), PP. 37-48, 2013.
- [6] Ferziger, Joel H., Peri, C. Milovan, "Computational Methods for Fluid Dynamics", 3<sup>rd</sup> revised edition, Springer-Verlag Berlin Heidelberg New York 2002.