Vol.10, Issue No 3, Sep 2020

Exhaust Heat Exchanger Thermodynamic Modeling in Gas-Fired Power Plants

Dr.Pratap.S.Kulkarni,Suresh Patil.G.L,A.Girish

Assoc. Prof, Asst. Prof, Asst. Prof

spratapkulkarni@gmail.com, glspatil@gmail.com, girish.hospet@gmail.com

Department of Mechanical, Proudhadevaraya Institute of Technology, Abheraj Baldota Rd, Indiranagar, Hosapete, Karnataka-583225

ABSTRACT

We need reliable and affordable energy sources if we want our economy to grow. Unfortunately, our current energy sources are dwindling at an alarming pace. That is why we need new ways of thinking about energy efficiency. This work aims to introduce a ceramic heat exchanger with a novel cross-section architecture.Issues with inadequate mass distribution and thermal tensions induced by uneven heating and cooling were among the several challenges that early heat exchangers set the way for in terms of process intensification. Some have suggested using ceramic materials instead of more traditional ones to circumvent these issues. Ceramics are a great option because of all the great things they can do, such being resistant to heat and corrosion. The meticulously crafted surfaces of ceramic heat exchangers provide a uniform dispersion of incoming fluxes.

Computer fluid dynamics (CFD) were used in this study to model various ceramic heat exchanger tube layouts.The real model of the complex, multi-shaped building was imported using Fluent 18.2. The ceramic monolith heat exchanger was designed by engineers to test how well heat is transferred. The whole domain, including the air-side fluid region, the ceramic core, and the exhaust gas side, was subjected to numerical simulations. With air and exhaust moving in opposite directions, the entire system was determined using a variety of duct cross-sections, including rectangular, elliptical, and cylindrical. A 15% increase in the predicted heat transfer rate is produced by numerical simulations, as compared to theoretical predictions.

INTRODUCTION

Conserving energy now requires more work.There is a scarcity of energy, in large part because fossil fuels, which are essential to its production, are in short supply. Oil, natural gas, and other energy resources are among many that are at our disposal. More energy is required to power modern automobiles. If production keeps up at its current rate, oil reserves will be exhausted soon. Nevertheless, if production rates start to decline, the limited availability of fossil fuels will pose a greater threat.There is an even more critical need to develop vehicles that use less energy in order to reduce emissions of greenhouse gases. Because they soak up heat reflected back from the Earth's surface, emissions of carbon dioxide (CO_2) from burning fossil fuels contribute to global warming. Because the greenhouse effect disrupts the natural marine and carbon cycles, the environment's capacity to store CO2 is reduced. The burning of fossil fuels for energy generation accounted for 45 percent of the world's CO_2 emissions. Total emissions of carbon dioxide (CO2) from human activities have increased by nearly 80% since 1970, with the transportation sector accounting for almost 100% of that increase, leading to an increase of around 0.8 degrees Celsius in average atmospheric temperatures. On a larger scale, this may not seem significant, but this tendency is predicted to have disastrous consequences over the world. As a result of the pressing need to reduce emissions and deplete fossil fuel sources, there is a strong focus on environmentally friendly engine and vehicle development. Thus, it's clear that electric automobiles should lessen the CO2 emissions from renewable sources. Reducing these losses has been a major focus in the quest to make combustion engines more efficient. To do this, it is possible to enhance the combustion efficiency of the engine, reduce losses due to mechanical friction, and build better gas exchange paths.

1.1 OBJECTIVES OF PRESENT STUDY

- The present work include following objectives
- Designing of Rectangular, circular and elliptical shapes using CREO-PARAMETRIC 3.0
- Simulating the designs with ANSYS FLUENT 15.
- To enhance heat transfer coefficient of Rectangular, circular and elliptical tubes.
- Calculating the heat transfer rate.
- 5. Calculation of heat transfer coefficient for Rectangular, circular and elliptical shape tubes
- Comparing heat transfer coefficient between optimized shapes of tubes.

1. LITERATURE REVIEW

Young Hawn Yoon[1](2009) In their study they found the performance of heat transfer and pressure drop by numerical computation and ξ-NTU method. By comparisons of both performances by the numerical computation and the ξ-NTU method, the effectiveness by ξ-NTU method was closest to the result by the numerical computation within the relative of 2.14%when Stephan's Nusselt number correlation was adopted to the ξ-NTU method among the several correlations.

Chandrakala[2](2015)- In their study the performance of heat transfer and pressure drop is calibrated by numerical computation. The main aim is to reduce the hot side temperature from 1100oc to 600oc and later it passes through the metallic heat exchanger temperature ranges less than 600oc. By increasing the Reynolds number on the cold air side this increase the velocity of the cold fluid Increase the heat transfer rate also increase the velocity by using nuzzling effect on cold air slot. The main purpose using the ceramic is to withstand with high temperature than metal.

P.Sowjanya[3](2016);In their study they stated that Ceramic heat exchanger has low material cost and also it can withstand high temperatures compared to metallic heat exchanger. Due to this reason it is important to predict the performance of ceramic heat exchanger, before it gets fabricated .In this project CFD analysis is performed on the ceramic heat exchanger having rectangular and circular ducts where aluminum nitride is used to predict and optimize various parameters like heat transfer rate and effectiveness.

2. WORKING OF CFD CODE:

All the CFD codes contain three main elements. They are as follows,

- Pre-processor.
- Solver.
- Post processor.

3.3.1 Pre Processor

It transfers the input of a flow problem to CFD program by means of an operator friendly interface and the subsequent transformation of this input into a suitable format, which can be used by solver. The stage wise preprocessor activities include.

- Determining the geometry of the region of the interest i.e. the computational domain.
- Grid generation or mesh generation (subdivision of the computational domain into small segments, which are called as cells, control volumes)
- Selection of the physical and chemical phenomena that need to be modeled.
- Definition of fluid properties.
- Specification of appropriate boundary conditions at cells, which coincide with or touch the domain boundary.

The solution to a flow problem (pressure, velocity, temperature etc.) is defined at nodes, corners of each cell. The number of the cells in the grid governs the accuracy of a CFD solution. In general, the larger number of cells the better the solution accuracy, but increases the time required for solution.

3.3.2 Solver

There are three distinct of numerical solution techniques: finite difference, finite element and finite volume method. The outlines of the numerical method that form the basis of the solver perform the following sequence steps:

- Approximation of the unknown flow variables by means of simple function.
- Discretization by substitution of the approximation into the governing equations and subsequent mathematical manipulation.
- Solution of the algebraic equation through an interactive process.

3.3.3 Post Processor

As in preprocessing, a huge amount of development work has recently taken place in the post-processing field. Owing to the increased popularity of engineering workstations, many of which have outstanding graphics capabilities, the leading CFD package are now equipped with versatile data visualization tools. These includes,

3. ANALYSIS OF HEAT EXCHANGER

4.1 Assumptions

1. The governing equations are assumed to be in steady state and taken for compressible fluid.
2. The fluid flowing through the heat sink channel exhibits Newtonian behaviour.
3. The density of the air is taken at constant pressure and at ambient temperature
4. Inlet velocity and temperature of the rectangular and trapezoidal heat sinks isuniform.
5. Uniform air velocity is assumed along the length of the fin
6. The wall resistance and fouling factors are negligible.
7. All the heat rejected from microelectronic processing system assumed to be absorbed in heat sinks.

4.4NUMERICAL ANALYSIS OF HEAT EXCHANGER

4.4.1. Geometrical Model of rectangular tube heat exchanger;

The geometric model for the rectangular tube Heat Exchanger is as shown in the Fig. 4.2.1

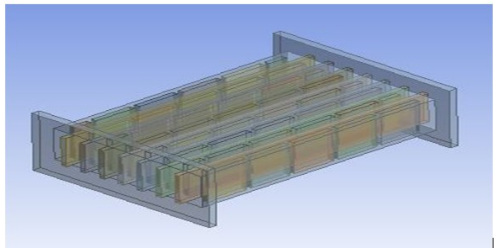

Fig.4.4.1 Fig shows Geometric Model of rectangular tube heat exchanger.

4.4.2. Meshing module of rectangular tube heat exchanger;

The meshing module for the rectangular tube Heat Exchanger is as shown in the Fig. 5.4.2

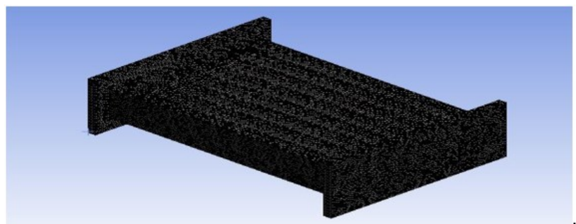

Fig.4.4.2 shows meshing module of rectangular tube heat exchanger.

4.4.3temperature,Pressureandvelocity distributions of rectangular Structure:

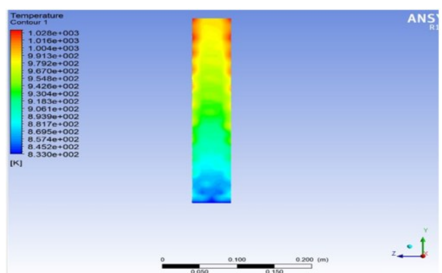

Fig. 4.4.3 Shows Contours Of temperature for rectangular tube heat exchanger at air side

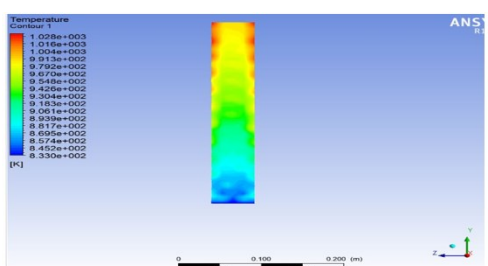

Fig. 4.4.4 Shows Contours Of velocity for rectangular tube heat exchanger at exhaust side

TEMPERATURE, PRESSURE AND VELOCITY DISTRIBUTIONS OF ELLIPTICAL STRUCTURE:

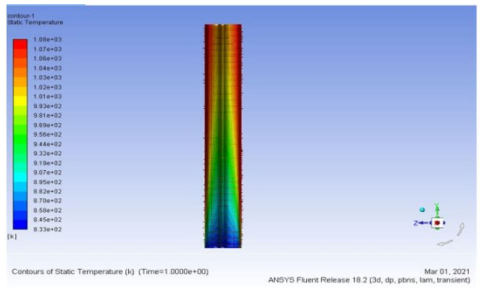

Fig. 4.3.3. Shows Contours Of temperature for elliptical tube heat exchanger at air side

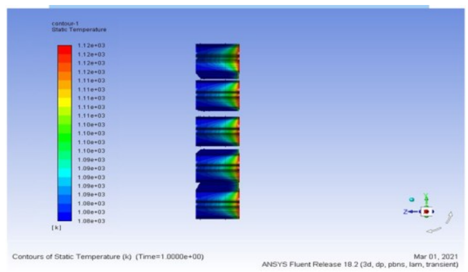

Fig. 4.3.4 Shows Contours Of temperature for elliptical tube heat exchanger at exhaust side

4. Results &Discussion

GRAPHICAL REPRESENTATION:

Fig.5.1. mass flow rate vs. heat transfer rate rectangular tube:

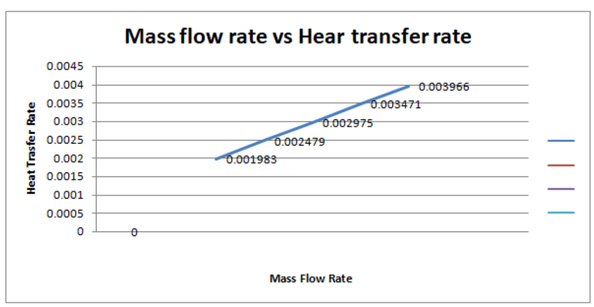

FIG.5.1. mass flow rate VS heat transfer rate

The above Fig.5.1 shows the variation of heat transfer rate with the mass flow rate. As the mass flow rate is increases heat transfer rate is also increases With respect to the mass flow rate. Mass flow ratewill changes heat transfer rate either rise or fall down and then maximum heat transfer rate will be obtained at 0.003966kg/s.

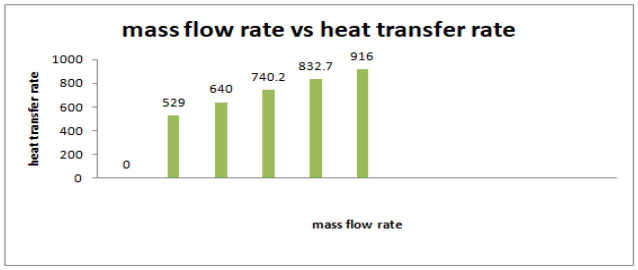

FIG.5.2. mass flow rate VS heat transfer rate

The above Fig.6.2 shows the variation of heat transfer rate with the mass flow rate. As the mass flow rate is increases heat transfer rate is also increases With respect to the mass flow rate. Mass flow rate will changes heat transfer rate either rise or fall down and then maximum heat transfer rate will be obtained at 0.003966kg/s of an elliptical tube

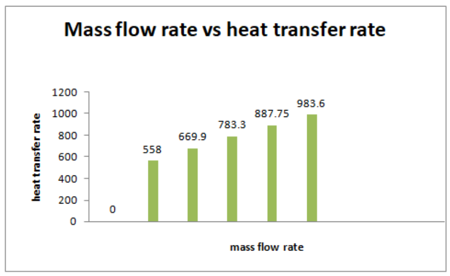

FIG.5.3. mass flow rate VS heat transfer rate

The above Fig.6.3 shows the variation of mass flow rate with heat transfer rate. As the mass flow rate is increases the heat transfer rate is also increases.The above Fig.6.1 shows the variation of heat transfer rate with the mass flow rate. As the mass flow rate is increases heat transfer rate is also increases With respect to the mass flow rate. Mass flow rate will changes heat transfer rate either rise or fall down and then maximum heat transfer rate will be obtained at 0.003966kg/s.

5.6 RESULT TABULAR

Comparison between Reynolds Number & Correlations

Reynolds No		585	736	888	1040	1192
Rectangle	Pressure	0.0198	0.0169	0.00921	0.00650	0.00234
	Velocity	6.231	4.934	2.421	1.0747	0.0543
	Temperature	835.2	834.5	833.12	832.97	831.59
Elliptical	Pressure	0.018064	0.01690078	0.01323807	0.006509	0.00356
	Velocity	0.61700254	0.77143	0.92246	1.0747096	1.098563
	Temperature	833.34	833.15	833.03	832.97	832.56
Cylinder	Pressure	0.227896	0.249018	0.262542	0.293062	0.318870
	Velocity	0.098198	0.196825	1.17370	1.355605	1.5821868
	temperature	832.34	831.01	830.12	829.46	829.16

Comparison between Mass flow rate & Correlations

Mass flow rate	0.001983	0.002479	0.002975	0.00347	0.003966
Reynolds no	585	736	888	1040	1192
Rectangle	525	636.29	736.9	828.86	912.137
Elliptical	529	640	740.2	832.7	916
cylindrical	558	669.9	783.3	887.75	983.6

Effectiveness between rectangle,elliptical and cylindrical shapes from numerical analysis

Comparison between theoretical and numerical effectiveness						
Reynolds no		585	736	888	1040	1192
Effectiveness(Theoretical)	Elliptical	0.41	0.42	0.43	0.45	0.46
	Cylindrical	0.54	0.56	0.57	0.59	0.6
	Rectangle	0.53	0.54	0.55	0.57	0.58
Effectiveness(Numerical)	Elliptical	0.56	0.56	0.57	0.6	0.61
	Cylindrical	0.69	0.71	0.72	0.74	0.75

5. CONCLUSION

The efficiency of a ceramic monolith heat exchanger was assessed both theoretically and numerically in this study. Exhaust gas and ambient air at temperatures between 600 and 1000 C have been the subject of theoretical study. The overall effectiveness and pace of heat transfer were assessed by theoretical and numerical investigations.There has been heavy reliance on Nusselt number correlations found in the literature in the NTU method computations. (1) Compared to the theoretical study, the numerical simulation showed a 15% improvement in heat transfer performance. Comparing the numerical approach to finding the Nusselt number to the total heat transfer calculated using the NTU technique with the Stephen correlation yields the most similar results.

Two, various shapes of heat exchanger tubes have varying anticipated efficiencies: rectangular tubes have 52% efficiency, elliptical tubes have 55% efficiency, and cylindrical tubes have 62% efficiency.

6. FUTURE SCOPE:

In this research, the ceramic materials employed are Ni-Cr-Al, NiCrAl + $MgZrO_3$, and $MgZrO_3$. Similarly, cutting-edge ceramic materials with inherent heat resistance might be used to do the same.

Different cross sections than the usual rectangular, elliptical, and cylindrical ones may be used as long as the mass distribution remains the same and the L/D ratio is correct.

The smoke stack exhaust might be redirected to preheating the air in the heater instead of being vented outside, enabling the heater to operate at a lower and more energy-efficient temperature. Assuming you know what application settings to use, this is totally doable.

REFERENCES:

Theoretical Investigation and Computational Fluid Dynamics Modeling of the Ceramic Monolith Heat Exchanger [1] (17-21-2009) Citation: Yoon Young Hwan1, Paeng Jin Gi2, and Kim Ki Chul 3.

A Ceramic Monolith Heat Exchanger: A Theoretical Investigation and Performance Evaluation [2]

T. Karthikeyan, N. Prabhu, and M. Dev Anand1* are the authors of this work. Year: 2009.

Examining Improved Tubes in a Dual-Pipe Heat Exchanger to Compare Their Flow and Other Features Antony Luki Ganesan Jr.A(1).M equals two.

In International Communications in Heat and Mass Transfer 36 (2009), pages 698-704, you may find the article "Compound heat transfer enhancement of a dimpled tube with a twisted tape swirl generator" written by Chinaruk Thianpong a, Petpices Eiamsa-ard a, Khwanchit Wongcharee b, and Smith Eiamsa-ard

Publication: "Development and evaluation of enhanced heat transfer tubes," volume 31, issue 11, pages 2141-2145, publishing year: 2011, Kevin G. Fuller (a), Rick Smith (b), and David J. Kukulka (a).

In their 2001 article "Heat transfer enhancement in dimpled tubes," published in Applied Thermal Engineering, Juin Chen (a), Hans Muller-Steinhagen (b), and Georey G. Ducy (a) [6] state that dimples improve heat transport.

J.H. Dooa, C. Son, M.Y. Haa, H.S. Yoonb, and J.E. Kima [7]. The 2012 International Journal of Heat and Mass Transfer article "Numerical investigation on flow and heat transfer characteristics in a cooling channel with a protrusion-in-dimple surface" (55, 7257-1026) addresses this topic.

In "Flow and Heat Transfer Performance of A Dimpled-Inter Surface Heat Exchanger—an Experimental /Numerical Study" [8], M. A. Saleh et al. establish the following. Chapter 2, Page 21 of Thermal Engineering, 2002.

Pedro G. Vicente, Alberto Garcia, and Antonio Viedma published an article in the International Journal of Heat and Mass Transfer in 2002 titled "Heat transfer and pressure drop for low reynolds turbulent flow in helically dimpled tubes." The article can be found on pages 543-553.

*Peter G. Vicente, Alberto Garca, and Antonio Viedma may be found on page 10. The paper titled "Experimental study of mixed convection and pressure drop in helically dimpled tubes for laminar and transition flow" was published in the International Journal of Heat and Mass Transfer in 2002 and can be found on pages 5091-5105.