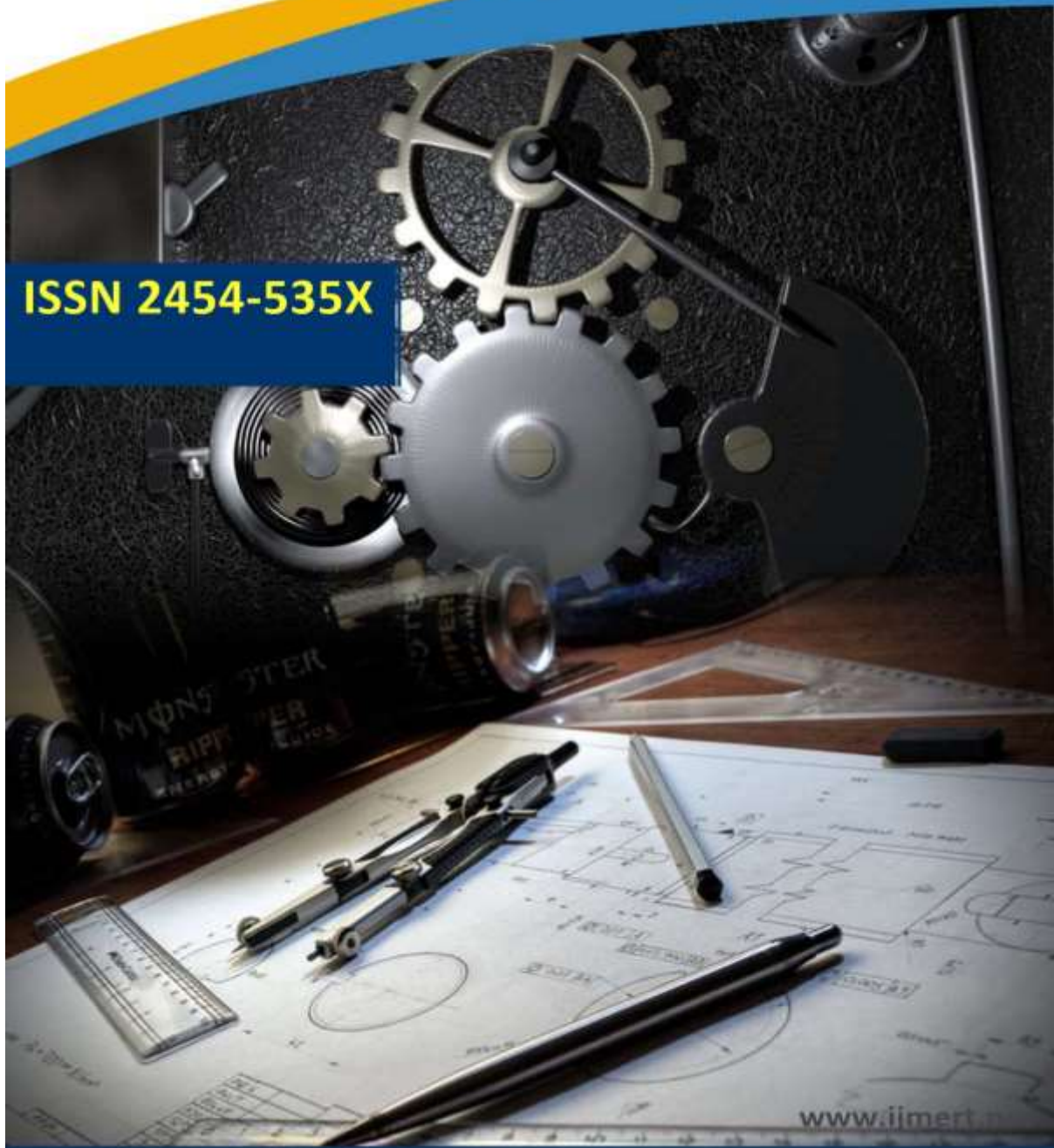




International Journal of
Mechanical Engineering Research and Technology

ISSN 2454-535X



Email ID: info.ijmert@gmail.com or editor@ijmert.net



Gas Exhaust Heat Exchanger CFD Study of Heat Recovery in a Gas-Fired Power Plant

A Hari Krishna¹, S Vijay Kumar², B Srihari³, J Suresh Babu⁴

ABSTRACT

The availability of affordable, reliable energy sources is crucial to a thriving economy. However, the current energy sources are being used up at an alarming pace. Therefore, new methods of energy conservation must be implemented. This is an effort to market a ceramic heat exchanger with novel cross sections. Inadequate mass distribution, thermal stresses due to inhomogeneous heating and cooling, etc. are only some of the many process intensification challenges that early heat exchangers helped pave the way for. Ceramic materials have been presented as a replacement for traditional materials as a means of overcoming the aforementioned challenges. Given that ceramic materials are advantageous in many ways, including their resistance to corrosion and high temperature. Ceramic heat exchangers' specially designed surfaces ensure that incoming fluxes are distributed uniformly.

Computational fluid dynamics (CFD) was used to model a variety of ceramic heat exchanger tube configurations for this study. The physical model of the complex, multi-shaped construction was brought into Fluent 18.2. To measure how efficient and effective heat transmission is, engineers developed the ceramic monolith heat exchanger. The whole domain, including the fluid area on the exhaust gas side, the ceramic core, and the fluid region on the air side, was computed numerically. Different duct cross sections (rectangular, elliptical, and cylindrical) with air and exhaust flowing in opposite directions were used to calculate the total system. When the predicted heat transfer rate is compared between theoretical calculation and numerical computing, the latter is shown to be 15% higher.

INTRODUCTION

These days, it takes more effort to save energy. Due in large part to a lack of fossil fuels, which are used in the production of energy, there is a restricted supply of such energy. There are a wide variety of energy resources available, including oil, natural gas, and others. Today's vehicles are more energy-hungry than ever. Oil reserves will be depleted shortly if production continues at its present pace. The finite supply of fossil fuels, however, will become more problematic when production rates begin to drop.

The release of greenhouse gases is another, more pressing reason to create energy-efficient automobiles. Emissions of carbon dioxide (CO₂) from the burning of fossil fuels contribute to global warming because they absorb heat reflected back from Earth's surface. The ability of the environment to store CO₂ is diminished as a result of the greenhouse effect's interference with natural marine and carbon cycles. Energy production from the combustion of fossil fuels was the single greatest contributor to global CO₂ emissions (45 percent). Since 1970, human-caused carbon dioxide (CO₂) emissions have grown by around 80 percent overall (and by more than 100 percent in the transportation sector), raising average atmospheric temperatures by about 0.8 degrees Celsius. While this may not seem like much on a grand scale, the global impact of this trend is expected to be catastrophic.

^{1,2,3,4} Asst. Professor,

Department of ME, K. S. R. M College of Engineering(A), Kadapa



There is a significant emphasis on environmental sustainability in vehicle and engine development due to the need to cut emissions and decrease the use of fossil fuel reserves. Therefore, it is evident that the CO₂ emissions from renewable sources should be reduced by the use of electric cars.

There has been significant effort put towards improving the efficiency of combustion engines by decreasing these losses. The losses caused by mechanical friction can be decreased, the engine's combustion efficiency can be improved, and better gas exchange pathways may be designed to accomplish this goal.

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 is uniform.
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.

4.4 NUMERICAL 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

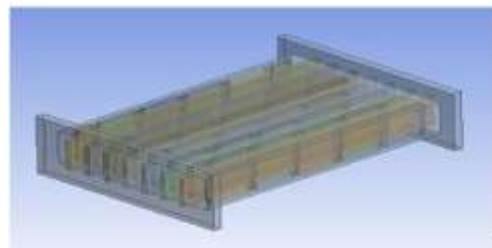


Fig.4.4.1 Fig shows Geometric Model of rectangular tube heat exchanger.

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

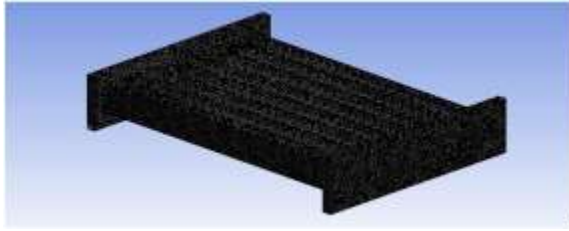


Fig.4.4.2 shows meshing module of rectangular tube heat exchanger.

4.4.3 temperature, Pressure and velocity distributions of rectangular Structure:

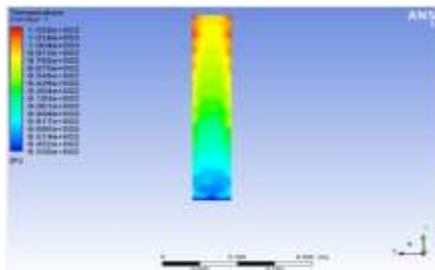


Fig. 4.4.3 Shows Contours Of temperature for rectangular tube heat exchanger at air side

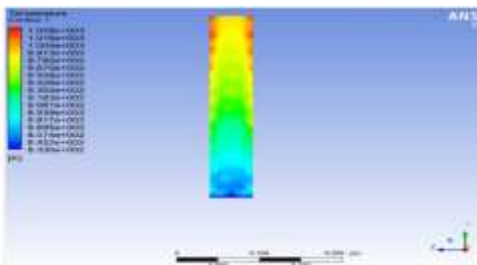


Fig. 4.4.4 Shows Contours Of velocity for rectangular tube heat exchanger at exhaust side

TEMPERATURE, PRESSURE AND VELOCITY DISTRIBUTIONS OF ELLIPTICAL STRUCTURE:

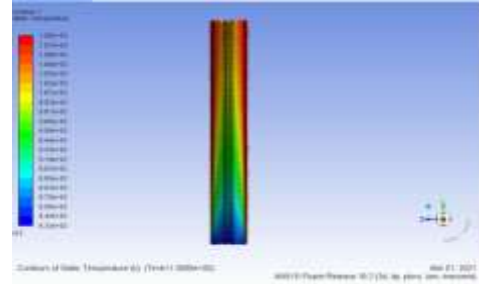


Fig. 4.3.3. Shows Contours Of temperature for elliptical tube heat exchanger at air side

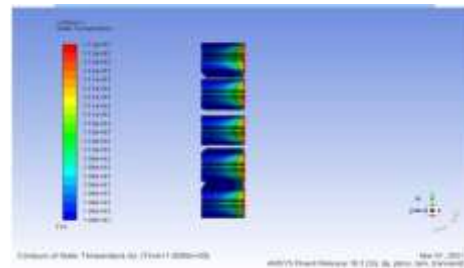


Fig. 4.3.4 Shows Contours Of temperature for elliptical tube heat exchanger at exhaust side

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 rate will changes heat transfer rate either rise or fall down and then maximum heat transfer rate will be obtained at 0.003966kg/s.

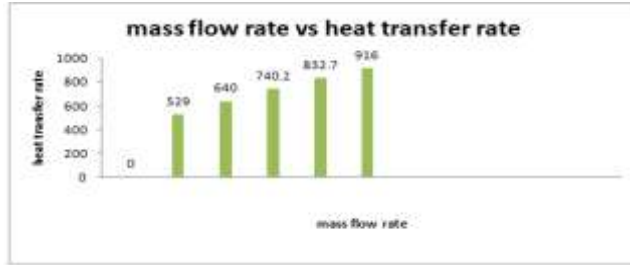


FIG.5.2. mass flow rate VS heat transfer rate

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

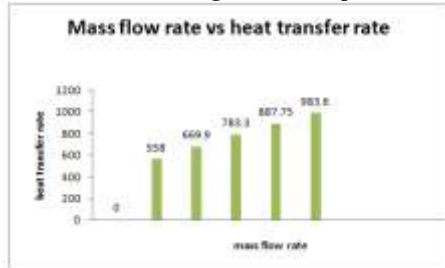


FIG.5.3. mass flow rate VS heat transfer rate

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	0.0208	0.0189	0.0092	0.0050	0.0038
	Velocity	0.231	0.294	0.421	0.477	0.504
	Temperature	815.2	818.5	818.2	812.97	815.36
Elliptical	Pressure	0.018084	0.0159818	0.0133097	0.009599	0.0076
	Velocity	0.05780254	0.77141	0.92846	1.3747985	1.088563
	Temperature	815.34	815.18	813.81	812.97	812.36
Cylinder	Pressure	0.227986	0.189928	0.262542	0.298861	0.338819
	Velocity	0.098188	0.196251	0.27779	0.388805	1.0421888
	Temperature	812.34	811.31	818.22	825.48	815.18

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	838.86	912.137
Elliptical	529	640	740.2	832.7	916
Cylindrical	558	669.9	788.3	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 present study contrasted theoretical and numerical analyses of the performance of a ceramic monolith heat exchanger. Exhaust gas and ambient air at temperatures between 600 and 1000 C have been the subject of theoretical studies. Theoretical and numerical analyses assessed the overall heat transfer rate and efficacy. Numerous Nusselt number correlations found in the literature have been used in the computations that have been carried out using the NTU approach.

(1) Numerical calculation improved heat transfer performance by 15% compared to theoretical study. The total heat transfer by NTU technique with the Stephen correlation is the most similar to the numerical approach of calculating the Nusselt number.

(2) Rectangular tube heat exchangers have an estimated Efficiency of 52%, elliptical tube heat exchangers have an Efficiency of 55%, and cylindrical tube heat exchangers have an Efficiency of 62%.

6. FUTURE SCOPE:

Ni-Cr-Al, NiCrAl + MgZrO₃, and MgZrO₃ are the ceramic materials used for the whole of this project. The same may be achieved by using state-



of-the-art ceramic materials that are naturally resistant to high temperatures.

By keeping the right L/D ratio and without changing the mass distribution, you may employ cross sections other than the standard rectangular, elliptical, and cylindrical ones.

As an alternative to venting smoke stack exhaust outside, it may be used to preheat air in the pre heater, allowing it to function at a lower, more energy-efficient temperature. This is possible so long as the appropriate parameters for the application are known.

REFERENCES:

The Ceramic Monolith Heat Exchanger: Theoretical Analysis and Computational Fluid Dynamics Simulation [1]

(17-21-2009) Young Hwan Yoon¹, Jin Gi Paeng², and Ki Chul Kim³.

A Theoretical Study and Performance Analysis of the Ceramic Monolith Heat Exchanger [2]

Authors: M. Dev Anand^{1*}, G. Glan Devadhas², N. Prabhu³, and T. Karthikeyan⁴. Year: 2009.

Analysis of Enhanced Tubes in a Double-Pipe Heat Exchanger for Flow and Characteristics Comparison Ganesan, Antony luki.A(1).M(2) .

Read "Compound heat transfer enhancement of a dimpled tube with a twisted tape swirl generator" by Chinारuk Thianpong a, Petpices Eiamsa-ard a, Khwanchit Wongcharee b, and Smith Eiamsa-ard c in International Communications in Heat and Mass Transfer 36 (2009), pages 698-704.

"Development and evaluation of enhanced heat transfer tubes," Applied Thermal

Engineering 31 (2011) 2141-2145, David J. Kukulka a, Rick Smith b, Kevin G. Fuller b.

"Heat transfer enhancement in dimpled tubes," Applied Thermal Engineering 21 (2001) 535-547, Juin Chen (a), Hans Muller-Steinhagen (b), and Georey G. Ducey (a) [6].

[7]. C. Son, J.E. Kim, J.H. Dooa, M.Y. Haa, H.S. Yoonb, and J.H. Dooa Numerical investigation on flow and heat transfer characteristics in a cooling channel with a protrusion-in-dimple surface, International Journal of Heat and Mass Transfer, 55, 7257-7267, 2012.

According to M. A. Saleh et al.'s "Flow and Heat Transfer Performance of A Dimpled-Inter Surface Heat Exchanger-an Experimental/Numerical Study" [8]. Thermal Engineering, Volume 21, Issue 2, 2002).

Heat transfer and pressure drop for low reynolds turbulent flow in helically dimpled tubes, International journal of heat and mass transfer 45 (2002) 543-553, Pedro g. vicente, Alberto Garcia, and Antonio Viedma.

On page 10, you'll find Pedro G. Vicente *, Alberto Garca, and Antonio Viedma. International Journal of Heat and Mass Transfer 45 (2002) 5091-5105, "Experimental study of mixed convection and pressure drop in helically dimpled tubes for laminar and transition flow."

