Engineering Universe for Scientific Research and Management
ISSN (Online): 2319-3069
Vol. XVII Issue V
May 2025

A Review on Computational Analysis of Heat Transfer Performance of Gasifier Fins using FEM

Vikash Kumar Choube[1] , Dr. DS Rajput[2]
PG Scholar [2]Assistant Professor[1]
Department of Mechanical Engineering[1,2]
Sri Satya Sai University of Technology and Medical Sciences Sehore, Bhopal (MP) [1,2]

Abstract

Carburetors are used to manufacture gas, and pipes are used to transport gas from the gas fur to the IC engine. Flin is an extended surface that emerges from the surface or body and increases the heat transfer rate between the surface and the surrounding liquid by increasing the heat transfer range. Carburetor power plants use condensation elements to reduce the temperature of hot gases discovered by carburetors. In this condensation element, fins are used to reduce the temperature difference.

We numerically examined fixed convection heat transfer from rectangles, triangles, and proposed fins extending vertically from the embarrassing rectangular base. The effects of geometric parameters and fundamental ambient temperature differences on the heat transfer performance of the FIN array were observed to determine the optimal FIN separation value. And in this current wok, heat flow, temperature distribution, and thermal gradients were discovered through simulation software. Comparison of calculation results and software results.

Keywords: ***Artificial Roughness, Solar Air Heater, Roughness Geometry, Nusselt Number, Friction Factor, Thermo Hydraulic Performance, Reynolds Number.***

1. Introduction

A fin is a surface that extends from an object to increase the speed of heat transfer within or from the environment by increasing convection. The height of the object's management, convection, or radiation determines the amount of heat it transmits. Meanwhile, in recent years, the issue of providing more efficient and reliable heating systems in terms of size, weight, cost and energy savings has been paid a great deal of attention from researchers. Many technical methods have been considered over the years to meet these requirements.

1.2 Method of Evaluation

Essentially there are three approaches or methods that can solve the problems of fluid flow and heat transfer. These approaches are:
1. Experimental approach
2. Theoretical or analytical
3. Numerical or Computer Technology

1.2.1 Experimental Approach

The most reliable information about physical processes is often given by actual measurements. An experimental approach using full-scale instrumentation can be used to predict how an identical copy of a device will do under the same conditions. In most cases, tests of such fully equipped equipment are unaffected and often impossible. An alternative is to run the experiment on a small model. However, the resulting information must be fully extracted, and the general rules for this are often not available. Furthermore, small models do not always simulate all the characteristics of a perfect instrument. Important features such as burning and cooking are often excluded from model testing. This further reduces the usefulness of the test results. Finally, it is important to remember that in many situations there are serious measurement difficulties and that the instrument is error free.

1.2.2 Theoretical Approach

Theoretical predictions satisfy the results of mathematical models rather than the results of actual physical models. For the physical process of interest, mathematical models mainly consist of many differential equations, along with related limitations and initial conditions. When classical mathematical methods are used to solve these equations, there is little hope to predict many phenomena of actual interest. Theoretical approach uses simplified assumptions to help you understand the problem.

1.2.3 Numerical Approach

Finite element methods or FEM are the analysis of systems that contain related phenomena such as liquid flow, heat transfer, and chemical reactions using computer-aided simulations. This technology is extremely powerful and includes a wide range of industrial and non-industrial application areas. Some examples are:

a) Aircraft and vehicle aerodynamics: lifting and resistance.

b) Ship fluid dynamics. Chemical Process Techniques: Mixing and Separation, Polymer Formation.

g) Sea development: Load on offshore structures.

The equations that regulate fluid flow are continuity (mass conservation), navigational stokes (impulse balance), and energy conservation (energy conservation). These equations form a system of combined nonlinear partial differential equations (PDEs). Due to the conjugation properties of equations and the presence of nonlinear terms, fluid flow equations generally do not have access to the analytical methods to obtain solutions. In general, analytical solutions are only possible if these PDEs can be created linearly by triggering nonlinear terms (as if they were in mucus or in irrational locations). If the nonlinearity of the major PDE cannot be ignored, this often requires the use of numerical techniques to obtain solutions in most technical rivers.

Compared to an experimental approach for fluid system design, FEM has several unique advantages.

a) Significant reduction in lead time and costs for new designs.

b) The ability to examine systems where controlled experiments are difficult or impossible to perform (e.g., very large systems)

c) The usual performance limitations (e.g., security research or accident scenarios)

d) The ability to examine systems under dangerous conditions under virtually unlimited levels of detail.

FEM code is composed mainly of numerical algorithms that can tackle liquid flow problems. To provide easy access to solution performance, all commercial FEM packages include a sophisticated user interface for entering problem parameters and examining the results. Therefore, all code contains three main elements: A brief look at the functionality of each of these elements in the context of FEM code with (i) a spare processor, (ii) a lover, and (iii) a post processor.

2. Literature Survey

2.1 Introduction

Heat treatment methods have played an important role in improving material properties. Train tests were conducted to investigate mechanical properties (hardness, ductility, tensile strength, etc.). Many tests have been carried out to restore mechanical properties for heat treatment, as shown below.

2.2 Previous Study

P. Sai Chaitanya et al. [1] Thermal analysis of engine cylinder fins was investigated. It is helpful to know the thermal department in the cylinder. The main purpose of the current work is to analyze the thermal properties through various shapes, materials and thicknesses of cylinder fins using ANSYS workbanks. Temporary thermal analysis determines the temperature and other thermal sizes that change over time. The fluctuations in temperature distribution over time are of interest for many applications, such as cooling. Accurate thermal simulations allow you to identify key design parameters with improved lifespan. The material A204 aluminum alloy used in the production of cylinder fin bodies has a thermal conductivity of 110-150 W/MK. Currently, cylinder fin analysis is carried out using this material and also using aluminum alloy 6061, which has high thermal conductivity.

Bharti Sharma et al. [2] used ANSYS as a tool to observe the geometric examination of FINS. The purpose of this study is to stop the performance of the fins using different geometry circles, rectangular, trapezoidal, and fin holes. Through ANSYS simulations of FINS under both conditions, with or without holes, this study seeks to analyze the heat transfer rate of FINS. Our current efforts focus on improving efficiency. This project will focus on examining ways to improve the performance of fins with or without holes.

Deepak Gupta et al. [3] it has been found that effective cooling can improve the life and effectiveness of the motor. The main goal of this project is to compare, compare and compare thermal properties with different geometry, materials and thicknesses with 100 cm Hero Honda Motorcycle Fins. Parametric cylinder models with FINS were developed to predict temporary thermal behavior. Models are generated by varying geometry such as rectangular, circular fins and varying fins to 3 mm and 2.5 mm. The 3D modeling software used is Pro/Engineer. The analysis is carried out in ANSYS. The material used in the production of the model of gray cast iron with a thermal conductivity of 53.3 w/mk aluminum alloy has a thermal conductivity of 200 W/mk. Analyze the designed model by taking over a thermal temperature of 11,000°C

Shekhar Shrivastav et al. [4] describes the paper on thermal analysis using finite element analysis element techniques to predict node shifts and tension in samples. To compare the output results, node shifts were analyzed using ANSYS software. Bars are structural elements characterized by preferred dimensions, their dimensions are both axial and length, and can also resist internal forces along their length. Currently, all machine members are considered cash and have different cross sections.

Komal Singh Rajput et al. [5] Numerical and analytical studies were conducted with the aim of optimizing the geometric fin parameters of Northbridge heat sinks used for natural convection heat transfer. It has been observed that perforated heat sinks specify a disruption in the thermal boundary layer that contributes to an increase in heat transfer rate. The main purpose of this work is to address defects by examining the effects of heat transfer rates in actual and interrupted heat sinks. **It** has also been observed that interrupted heat sinks also have fewer bulky agents containing fewer materials, thus reducing manufacturing costs. The use of a suspended design of the heat sink also indicates an increase in the total heat transfer rate from the surface. The results show that the proposed design in Northbridge's current work performs better compared to actual heat sinks.

Shekhar Shrivastav et al. [6] described a comparative study for thermal analysis of bar elements using finite elements. Finite Elements - Analysis is performed using CAD software (computer-aided design). The main purpose of the analysis is to investigate the behavior of the material against the thermal load of the material. This paper describes thermal analysis using finite element-analytical element techniques to reduce sample node shifting and tension. To compare the output results, node shifts were analyzed using ANSYS software. temperature values for both FEM and ANSYS software traditional equations for the proposed model.

Elnaggar et al. [7] presented the performance analysis of active U-shaped warm tubes used for desktop PC CPU cooling. The experiments are carried out by vertically mounting the system using a heat source in a rectangular tunnel, and power convection is facilitated using a blower. Thermal resistance (RT) and heat transfer coefficient are estimated in both forced convection modes under hospitalized patients' conditions by varying the thermal input of 4WTO 24 W and the air velocity from 1 m/s to 4 m/s. The experimental results and simulation results agree well.

Jong Bum Lee et al. [8] found that horizontal tubes with rectangular fins tilted are experimentally inspected for cooling of electronic devices. The temperature difference in horizontal tubes with tilted rectangular fins is measured for several thermal inputs, tilt angles, and fin counts. Nut prediction correlations have been proposed using measurement results. This correlation is suitable for situations with Rayleigh numbers of 200,000 1,100,000, tilt angles of 0 90, and fin counts from 9 36. Based on the correlation, different numbers of fins and thickness cooling services are displayed, and optimal cooling capacity values are found. Finally, we compare the optimum cooling capacity of tubes with rectangular fins and traditional radial rectangular fins. Comparison results show that the optimum cooling capacity of tubes with tilted fins is 6% higher than the optimum cooling capacity of tubes with radial rectangular fins. [9] presented an examination of the same rectangular fins (2, 4, 8, 10) as the hard rectangular aluminum fins, analytical, experimental, and validity through finite element analysis of temperature distribution along length. From this study, it has been observed that mathematics and FEA converges to 10 perforations within ±1 ëC and ±2 ·C, and therefore robust rectangular fins without perforations in rectangular fins with validity.

R. Arularasan et al. [10] shows the design of a single cylinder air cooling motor with different profile fins, such as triangle, rectangular, oval, and trapezoidal profiles, comparing existing profiles of the engine. Relative thermal inspection is performed using Aluminum-Zwei-s.i-i-motor. A cylinder with a single fin attached thereto for testing. Numerical simulations for the same setup were performed in ANSYS. The main goal of this work is to examine various previous tests to improve the heat transfer rate of the cooling fins by changing the cylinder block fin profile.

Bhopendra Dewangan et al. [11] investigated the behavior and effects of temperature through reactor pressure pipes. Finite difference method (FDM) and FEM software-Ansys are used to predict axial temperature distribution and impact on pressure tubes by including radioactive and convective boundary conditions. Results achieved with FDM and FEM software are well compared with experimental results. Prediction of the temperature distribution of cylindrical pressure tubes heated from the inside by coupling lines and radiation, and cooled from the outside by natural convection and radiation from the outside by external radiation, is specified in this article. The pressure tube is exposed to higher and lower temperatures at the bottom of the higher temperature. These two extreme temperatures are entered into the FDM and ANSYS software. The comparison is done with the experimental results, and the agreement between the mathematical model (FDM) and the ANSYS results is very good.

P Vijaya Sagar et al. [12] we decided to perform thermal analysis of the car cooler with or without relocated fins. This work uses ANSYS tools for computer analysis to perform CFD analysis of the cooler at different mass flow rates. Heat transfer analysis is performed to analyze heat transfer rates. The material used for colder fins is aluminum alloy 6061. Modeling is performed in Pro/engineer and analysis is performed in ANSYS.

Hossein Shokouhmand et al. [13] it was found that there is an optimal value for slander that can achieve maximum wall heat flow. The results also show important values for the coat of arms of the wall. The dependence of liquid to this ratio has almost disappeared. In most inspected cases, there is a greater improvement in heat transfer improvement in arc-shaped grooved channels as rectangular channels.

Farhanieh et al. [14] numerically and experimentally invest laminar flow and heat transfer properties in channels with rectangular grooved walls. Erresults showed improvements in local nuts compared to smooth parallel panel channels due to thermal boundary layer recovery and groove return formation. They also showed that this improvement was accompanied by a relatively high pressure increase. They showed the highest heat transfer rate, 45 degrees when combined promoter orientation. Their allegations showed an increase in heat transfer of 1.5 3.5 due to increased flow velocity in the grooved area compared to the basic grooved channel. However, we found that the decline in criminal waste has increased significantly compared to the previous year.

3. Objective & Methodology

3.1 Objectives

The main objectives of this study are:

i. Through constant state analysis, we find thermal effects (management and convection) in aluminum alloys.

ii. Compare the results of temperature distributions for different flower flows.

iii. Thermal gradients and heat flows in various aluminum alloys are determined and verified with analytical analysis.

A complete parametric model of FIN is created in the APDL software ANSYS APDL. The model obtained in step 2 is analyzed using ANSYS APDL to obtain heat rate, heating level, heat flow, and knot temperature. A manual calculation is performed. Finally, we compare the manual calculations with ANSYS results for different materials, shapes and thicknesses.

4. Governing Equations

4.1 Fin Equation

Consider a volume element of a fin at location x having length of x, cross-sectional area of A, and a perimeter of p, as shown in Fig. Under steady conditions, the energy balance on this volume element can be expressed as –

$$\begin{pmatrix}\text{Rate of heat}\\\text{conduction into}\\\text{the element x}\end{pmatrix} = \begin{pmatrix}\text{Rate of heat}\\\text{conduction fron}\\\text{the element x} + \Delta\text{x}\end{pmatrix} + \begin{pmatrix}\text{Rate of heat}\\\text{convection into}\\\text{the element}\end{pmatrix}$$

$$\frac{d^2\theta}{dx^2} = m^2\theta \text{, where, } m^2 = \frac{hp}{KA_c}$$

$$\theta(x) = C_1e^{mx} + C_2e^{-mx}$$

fin with finite length and tip un-insulated.

$$Q_{fin} = \sqrt{hpKAc}\,(T_S - T_a)\left(\frac{\tan h(ml) + \frac{h}{Km}}{1 + \frac{h}{Km}\tan h(ml)}\right)$$

$$\frac{\theta}{\theta_0} = \frac{T - Ta}{Ts - Ta} = \frac{\cos h\{m(l-x)\} + \frac{h}{Km}[\sin h\{m(l-x)\}]}{\cos h(ml) + \frac{h}{Km}[\sin h(ml)]}$$

Efficiency of fin (η_{fin}) =

$$\frac{\text{Actual heat transfer by the fin}}{\text{maximum heat that would be transferred if whole surface of the fin is maintained at the base temperature}}.$$

The heat generated from the source gets transferred by different modes that are conduction and convection; governing equations of heat transfer between the elements are applied. Below equation (1) represents the heat transfer which takes place during conduction derived by the Fourier, and equation (2) represents the heat transfer during conduction which is by Newton's law of cooling. The Fourier's Law of conduction – [11]

$$Q = -K.A_{c.}\frac{dt}{dx}$$

Newton's Law of convection –

$$Q = -h.(t - t_a)A_s$$

Where, Q is heat transfer rate, Ac is cross-sectional area in m^2, k is thermal conductivity (in W/m°C), h is Heat transfer coefficient (in $W/m^{2}°C$).

5. Finite Element Analysis

5.1 Introduction

Numerical analysis was performed to calculate the temperature and thermal voltage distribution of various flowers. The preferred system option for thermal analysis was selected. Numerical analysis was based on the following assumptions:

i) steady-state heat flow,

ii) The material is uniform and equal,

iii) The conventional heat transfer coefficient is the same everywhere.

5.2 FEM Implementations

1. Preferences (Peripheral Patient Status) â

Section §Select Fixed Condition Thermal Analysis from the Analysis System Toolbar. Pre-processing (creating FE models, meshes, BCs)

§Incoming analysis parameters and peripheral parameters.

3. Fem -solver (assembly and solve a system of equations)

§ Click Temperature for the solution. Click Solution. Click on the temperature distribution.
4. Post-processing (Sorting and Advertising Results Format)

5.3 Element Types

ANSYS APDL for analyzing gradients, tensions, and distractions of pipe/pipe walls by rivers, internal temperatures. This is because the tubes are exposed to internal temperature using three-dimensional elements (solid 278). However, increasing the number of elements generally means a more accurate result, but for a particular problem, finite elements are converted to a certain number of nodes. The number of nodes considered in this analysis was in the range 8000-10000. The following illustration shows the destruction of various flowers. The pipe material allows heat flow to be safely ignored, as only small vertical sections of the pipe were considered for analysis. Therefore, the top and bottom surfaces of the horizontal portion of the tube are isolated. The liquid mass temperature in the tube was maintained at 960°C and at 45°C for 1000 or 100 W/m2 K. To ensure bending resistance, each element of the tube (L) ratio (R) ratio (R) ratio is greater than 10. The heat flow was kept at zero at both pipe edges.

5.4 Solution:

To validate the thermal analysis results, calculations were performed and compared with ANSYS results by taking a minimum thickness value of 1-3 mm. This report provides maximum distraction, Atagawa, and thermal gradient distribution under operating temperatures. The purpose of this analysis is to ensure that the results of the finite element are compared with the results of the analysis and acceptance errors.

6. Scope for Future Work

In this work, a new type of fins provided and this design modeled through ANSYS APDL FE Software. The aim of the investigation consists into find out the effect of various shapes of conventional fins over new type of fins.

According to the results –

I. Fe simulation results showed that the temperature distribution of the proposed fins was the largest compared to different profiles (rectangular and triangular fins). The temperature distribution of the proposed fin increases by approximately 50%.

II. ANSYS - APDL simulation results determine that the river is the largest of the triangle profile and the smallest proposed profile, while the rectangular profile is between the proposed and triangle profiles. For the AA6061 class, the Thermal River is the largest compared to the AA2024.

III. It is examined for a decrease in heat flow along the fin type. If a rectangular profile is proposed, then we found that the river is the largest and minimal for the proposed profile, but the rectangular profile is between the proposed profile and the rectangular profile.

IV. We found that the proposed profile fins exhibit a lower chip temperature distribution than the others, like the table compared to the length of the fin and room optimization. According to the manufacturing aspects, the use of aluminum alloys 6061 and AA2024 should be considered. By changing the thickness of the FIN, the total manufacturing cost will further prepare new components.

References

[1]. Fahanieh, B., Herman, C., Sunden B., Numerical and experimental analysis of laminar fluid flow and forced convection heat transfer in a grooved duct, Int. J. Heat Mass Transfer, Vol. 36, 1993, pp. 1609-1617.

[2]. Bilen, K., Yapici, S., Heat transfer from a surface fitted with rectangular blocks at different orientation angle, Int. J. Heat Mass Transfer, Vol. 38, 2002, pp. 649-655.

[3]. Herman, C., Kang, E., Heat transfer enhancement in a grooved channel with curved vanes, Int. J. Heat Mass Transfer, Vol. 45, 2002, pp. 3741-3757.

[4]. O. Manca, S. Nardini, D. Ricci, A two-dimensional numerical investigation on forced convection in channels with transversal ribs, ASME Conference Proceedings,(43826), pp. 1099-1107,2009.

[5]. H. Shokouhmand, K. Valiidkhah, M.A. Esmaeili, Numerical analysis of air flow and conjugated heat transfer in internally grooved parallel-plate channels, World Acad. Sci. Eng. Technol. 73 (2011).

[6]. R. Kamali, A.R. Binesh, The importance of rib shape effects on the local heat transfer and flow friction characteristics of square ducts with ribbed internal surfaces, Int. Commun. Heat Mass Transf. 35 (8) 1032-1040, (2008).

[7]. P. Promvonge, C. Thianpong, Thermal performance assessment of turbulent channel flows over different shaped ribs, Int. Commun. Heat Mass Transf. 35 (10) 1327-1334, (2008).

[8]. C. Thianpong, T. Chompookham, S. Skullong, P. Promvonge, Thermal characterization of turbulent flow in a channel with isosceles triangular ribs, Int. Commun. Heat Mass Transf. 36 (7) 712-717, (2009).

[9]. D. Jansangsuk, C. Khanoknaiyakarn, P. Promvonge, Experimental study on heat transfer and pressure drop in a channel with triangular V-ribs, Energy and Sustainable Development: Issues

and Strategies (ESD), Proceedings of the International Conference on, pp. 1-8, IEEE 2010.

[10]. S. Eiamsa-ard, P. Promvonge, Numerical study on heat transfer of turbulent channel flow over periodic grooves, Int: Commun. Heat Mass Transf. 35 (7) 844-852, (2008).

[11]. S. Eiamsa-ard, P. Promvonge, Thermal characteristics of turbulent rib-grooved channel flows, Int. Commun. Heat Mass Transf. 36 (7) 705-711, (2009).

[12]. J. Cui, V.C. Patel, C.-L.Lin, Large-eddy simulation of turbulent flow in a channel with rib roughness, Int. J. Heat Fluid Flow 24 (3) 372-388, (2003).

[13]. A. Chaube, P.K. Sahoo, S.C. Solanki, Analysis of heat transfer augmentation and flow characteristics due to rib roughness over absorber plate of a solar air heater, Renew. Energy 31 (3) 317-331, (2006).

[14]. J.C. Han, Y.M. Zhang, C.P. Lee, Augmented heat transfer in square channels with parallel, crossed and V-shaped angled ribs, ASME, Journal of Heat Transfer 113590-596, (1991).

[15]. J.C. Han, Y.M. Zhang, C.P. Lee, Influence of surface heat flux ratio on heat transfer augmentation in square channels with parallel, crossed, and V-shaped angled ribs, ASME, journal of Turbo machinery 114 872-880, (1992).

[16]. J.C.Han, Y.M. Zhang, High performance heat transfer ducts with parallel broken and V-shaped broken ribs, International Journal of Heat and Mass Transfer 35513-523, (1992).

[17]. T.M. Liou, J.J. Hwang, Turbulent heat transfers augmentation and friction in periodic fully developed channel flows, ASME, Journal of Heat Transfer 114 56-64, (1992).

[18]. T.M. Liou, J.J. Hwang, Effect of ridge shapes on turbulent heat transfer and friction in a rectangular channel, International Journal of Heat and Mass Transfer 36 931-940, (1993).

[19]. S.W. Ahn, The effects of roughness types on friction factors and heat transfer in roughened rectangular duct, International Communications in Heat and Mass Transfer 28 (7) 933_942, (2001).

[20]. S. Mochizuki, A. Murata, M. Fukunaga, Effects of rib arrangements on pressure drop and heat transfer in a rib-roughened channel with a sharp 180° turn, ASME, Journal of Turbo machinery 119, 610-616, (1997).

[21]. G. Rau, M. Cakan, D. Moeller, T. Arts, The effect of periodic ribs on the local aerodynamic and heat transfer performance of a straight cooling channel, ASME, Journal of Turbo machinery 120 ,368-375, (1998).

[22]. A. Murata, S. Mochizuki, Comparison between laminar and turbulent heat transfer in a stationary square duct with transverse or angled rib turbulators, International Journal of Heat and Mass Transfer 44, 1127-1141, (2001).

[23]. K. Prasad, S.C. Mullick, Heat transfer characteristics of a solar air heater used fordrying purposes, Applied Energy 13, 83-93, (1983).

[24]. D. Gupta, S.C. Solanki, J.S. Saini, Thermo-hydraulic performance of solar air heaters with roughened absorber plates, Solar Energy 61 (1), 33-42, (1997).

[25]. J.L. Bhagoria, J.S. Saini, S.C. Solanki, Heat Transfer coefficient and friction factor correlation for rectangular solar air heater duct having transverse wedge shaped rib roughness on the absorber plate, Renewable Energy 25,341-369, (2002).

[26]. R. Karwa, Experimental studies of augmented heat transfer and friction in asymmetrically heated rectangular ducts with ribs on the heated wall in transverse, inclined, V.-continuous and V-discrete pattern, International Journal of Heat and Mass Transfer 30 (2) 241-250, (2003).

[27]. J.L. Bhagoria, M.M. Sahli, Augmentation of beat transfer coefficient by using 901broken ,rse ribs on absorber plate of solar air heater, Renewable Energy 25,2057-2073, trans' (2005).

[28]. Varun, R.P. Saint, S.K. Singal, A review on roughness geometry used in solar air heaters, Solar Energy 81, 1340-1350, (2007).