iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 8 **Issue:** VIII **Month of publication:** August 2020

DOI: https://doi.org/10.22214/ijraset.2020.30836

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 8 Issue VIII Aug 2020- Available at www.ijraset.com

Heat Transfer Analysis of Aluminum Section of a Fin & Tube Heat Exchanger

Vaddadi Rajendar[1], A. Murali Krishna[2], K. Ravindra[3], U. S. Ramakanth[4]
[1, 2, 3]*M. Tech Student, Department of Mechanical Engineering, #Visakha Technical Campus, Vizag, AP*
[4]*Associate Professor, Lendi Institute of Engineering and Technology, Vizianagaram, AP*

Abstract: Heat transfer analysis of fin & tube heat exchanger is majorly considered for cooling using convection process in most of the industrial process. In order to understand the temperature distribution and to minimize the heat losses, heat transfer analysis help us to understand the temperature profiles of fin & tube sections. An aluminum section of fin & tube heat exchanger with axi-symmetric model of 90 deg section is considered. The tube carries a contained hot fluid while the exterior surfaces experience a convective condition. It has been determined that the fluid causes a wall temperature of 200°C at the entrance and 150°C wall temperature at the exit. The effect of radiation is also considered which is very minimal significance is observed.
Keywords: Convection, Radiation, fin & tube heat exchanger, heat transfer

I. INTRODUCTION

Heat exchangers are widely used in many transportation applications, industrial or household, such as thermal power stations, heating media, transportation and air conditioning systems, electronic equipment and spacecraft. In all these applications, improving the efficiency of heat exchangers can lead to significant cost, space and material savings. Therefore, much research has been done in the past to find effective ways to improve the efficiency of heat exchangers. This research includes the selection of fluids with high-effective heat transfer surfaces made of high conductivity, high thermal conductivity materials and the selection of their flow arrangements. Effective techniques have been reported to improve heat transfer for both single and diode. However, in the current work, only the technique of stable natural convection was considered. The methods for improving the heat transfer in publications are summarized in several ways, but can mainly be grouped as active improvement methods. The basis of any technology to improve heat transfer lies in the use of some external energy to allow mixing of working fluids, rotation of heat transfer surfaces, vibration of heat transfer surfaces, or working fluids, as well as the generation of electrostatic fields. There are almost no industrial fields in which heat exchangers are not used. The design of heat exchangers greatly influences the design of the entire system or the process in which it is applied. Many factors affect the design of the heat exchanger, but the most important is the heat transfer rate. With the exception of a few cases, the high heat transfer rate and small pressure drop in small volume are generally required in all types of typical operations. Heat generally transmits in three basic ways: conduction, convection, and radiation. Density of thermal conductivity is not a difficult problem and can generally be controlled by materials chosen to construct the system. Additional radiation is of little importance when the heat transfer process occurs at moderate temperatures. The intensity of heat transmitted by convection is the predominant aspect of this type of analysis compared to conduction and radiation.According to Newton's cooling law, convection heat transfer can be calculated as a product of the heat transfer coefficient, the heat transfer surface area, and the temperature difference between the tube wall and the flowing fluid. Inside the walls. The temperature difference from wall to liquids is generally adjusted according to the operating conditions and therefore cannot be used to improve the heat transfer rate. You can increase the heat transfer surface area or heat transfer coefficient, or both at the same time. But since the heat transfer coefficient of a given material at a given temperature is constant, the only way to change the heat transfer rate is to change the surface heat transfer surface. Intermittent fins in the form of strips or mesh fins provide an increase in the heat transfer surface area as well as an increase in the effective coefficient of heat transfer. Therefore, these are especially effective in obtaining high rates of heat transfer. The mechanism that leads to the high heat transfer coefficients of these fins is the periodic interruption of the boundary layer around the fins, thus better mixing with fluid currents is achieved at different temperatures. Heat energy exchange is studied in a tube with a circular cross section and a specified internal and external radius with fins shaped like an outer disc. Finned tubes attached to the tube can be variable in shape and size. Three basic types of fins are considered and it is estimated to transfer heat energy from a tube with such configurations. Calculations of tube design and fin dimensions are made based on appropriate formulas for maximum heat transfer rate at low production costs.

The materials used in the calculations are ALUMINUM. The tube and fins are aluminum and the liquid inside the tube is water. ANSYS version 19.0 WORKBENCH uses all simulations. The experimental values of the working temperatures and the corresponding properties of the fin material and the tube with water are considered and sent to the program.

II. LITERATURE SURVEY

Cohen and Rohr [1] studied to direct mixed convection from a hot wavy surface. The channel flow between the nasal sinus surface and the flat top wall is examined using the combined DPIV and PLIF method to examine spatial changes with the current gap and normal wall velocity components, and to evaluate the synchronous field. From the tincture of the mark injected into the liquid. They found that the transport properties improved more compared to the mixed convection of the flat plate by the presence of the corrugated surface. Amador and others. [2] It was found that an improved heat transfer can be obtained without requiring higher volumetric flow rates in turbulent flow fluids, as more pumping forces are required. When the asymmetric wall channel operates in a suitable Reynolds number range, a significant improvement in heat transfer is observed. [3] A study of heat transfer and laminar flow in a three-dimensional corrugated microwave channel. It can be observed that the location and amount of eddies can vary along the direction of the flow, which leads to irregular flow, as the mixture of load liquids can be greatly improved and thus the heat transfer performance of the current corrugated micro channels is compared to the small straight straight channels with the same cross-section that the corrugation gives better Results. Lee et al. [4] It was observed in a fully developed flow experiment and heat transfer in intermittent corrugated channels with transverse rectangular regions thought to be using direct digital reproduction, to expand Reynolds numbers that cross from platelets to transient flow departments. It was found that due to the effective combination of the corrugated channels, the heat transfer performance is fundamentally better than that of the straight channels with the same cross-section; Meanwhile, the pressure drop penalty in corrugated channels can be much smaller than the heat transfer upgrade. Ramagadia and Saha [5] Simulation of fluid flow and heat transfer through a sinusoidal canal with a surface described by a sine wave function. Finally, thermal performance was investigated in Reynolds number 600 of various geometric shapes. Note that the pressure drop penalties due to the corrugated wall geometry decrease in the amplitude of the corrugation to a constant L / a ratio. The heat transfer by corrugated wall geometry is always greater than parallel plate composition. Naphon [6] presented the numerical results of the flow distribution and temperature in the channel with different corrugated plates of the engineering composition. The effects of the relevant parameter on flow structures and temperature are also considered. The sharp edge of the corrugated board can be found to have a great influence in improving heat transfer, especially the V-shaped corrugated board. Therefore, the use of corrugated board is a convenient way to increase performance.

More heat pressure for the heat exchanger. Muhammad and others. [7] The heat transfer properties and digitally searched water flow in the microscopic corrugated heat sink (WMCHS) ranged with a rectangular cross section with different corrugated capacities from 125 to 500 um. This research covers Reynolds number in the range from 100 to 1000. The field of water flow and heat transfer phenomena are simulated within the hot corrugated microscopic channels and the results are compared with straight microscopic channels. The effect of using a wavy flow channel on MCHS thermal performance, pressure drop, friction factor, and wall shear pressure has been reported in this article.

It was found that the heat transfer performance of small wavy channels is much better than small straight channels of the same cross section, and Nandy and Chattopadhyay digitally studied the development of irregular laminar flow and heat transfer at the same time at the same time. The micro-channel is wavy, due to the difference in velocity of the sinus component at the inlet. The flow was developing thermally and hydrodynamic while the canal walls were maintained at a uniform temperature. Simulation was performed in Prandtl plate number 7 and Reynolds number from 0.1 to 100.

Based on comparison with constant flow in the wavy channel it was found that the sine velocity at the inlet could provide improved heat transfer performance with different capacity (0.2, 0.5, 0.8) and frequency (1, 5, 10) Kuhn et al. [9] He studied the heat flow number of mixed water loads on a hot undulating surface over a range of Reynolds and Richardson numbers, including transitional and turbulent flow systems.

The integrated heat transfer of the corrugated wall formation was significantly improved (about 2.5 times) for Re = 1000, 2000 compared to the standard flat horizontal wall formation. Sui et al. [10] He conducted experimental research on flow friction and heat transfer in sinusoidal ducts with rectangular cross-sections. The fine channels considered consist of ten identical wavy units with an average width of about 205 mm, a depth of 404 mm, a wavelength of 2.5 mm, and a wave amplitude of 0-259 mm. Each test piece is made of copper and contains 60-62 microscopic channels undulating in parallel.

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 8 Issue VIII Aug 2020- Available at www.ijraset.com

III. HEAT TRANSFER AND ENHANCEMENT TECHNIQUES

The study of the transformation phenomenon, which includes the transfer of momentum, energy, mass, etc., has been identified as a unified system of fundamental importance dependent on flows and thermodynamic forces. The transmission of these phenomena is caused by the accompanying force of the temperature gradient, the speed gradient, the concentration gradient, the chemical affinity, etc. The transfer of heat energy due to the difference in temperature or gradient is called heat transfer. conduction refers to the transfer of heat between two bodies or two parts of the same body through molecules which are, more or less, stationary, as in the case of solids. The governing equation for conductive heat transfer is:

In Cartesian coordinates

$$\Delta f = \frac{\partial^2 f}{\partial x^2} + \frac{\partial^2 f}{\partial y^2} + \frac{\partial^2 f}{\partial z^2} = 0.$$

In cylindrical coordinates

$$\Delta f = \frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial f}{\partial r}\right) + \frac{1}{r^2}\frac{\partial^2 f}{\partial \phi^2} + \frac{\partial^2 f}{\partial z^2} = 0$$

In spherical coordinates,

$$\Delta f = \frac{1}{\rho^2}\frac{\partial}{\partial \rho}\left(\rho^2\frac{\partial f}{\partial \rho}\right) + \frac{1}{\rho^2 \sin\theta}\frac{\partial}{\partial \theta}\left(\sin\theta\frac{\partial f}{\partial \theta}\right) + \frac{1}{\rho^2 \sin^2\theta}\frac{\partial^2 f}{\partial \varphi^2} = 0.$$

Thermal radiation refers to the radiant energy emitted by bodies by virtue of their own temperatures, resulting from the thermal excitation of the molecules. Radiation is assumed to propagate in the form of electromagnetic waves. The governing equation for Radiation heat transfer is: PLANK'S LAW:

$$B_\nu(T) = \frac{2h\nu^3}{c^2}\frac{1}{e^{\frac{h\nu}{k_B T}} - 1}, \text{ or } B_\lambda(T) = \frac{2hc^2}{\lambda^5}\frac{1}{e^{\frac{hc}{\lambda k_B T}} - 1}$$

IV. NUMERICAL ANSLYSIS – STEADY STATE HEAT TRANSFER WITH CONVECTION CONDITION

The pre-processing, also called model preparation, is often the most work intensive step of the FEA. In the design module, the XY level for the engineering drawing is chosen and viewed. The unit is adjusted in millimeters and automatic restriction is activated. Six points are taken using building points and then combined with the help of different lines. From the Concept toolbar, sketch surfaces are taken, the previous drawing is applied and the surface is created and boundary conditions shown in Figure 1.1. In general, the unit of scale is taken mm because all dimensions are mm. The energy equation is activated and the laminar flow is taken. The water is taken as liquid and aluminum as a solid.

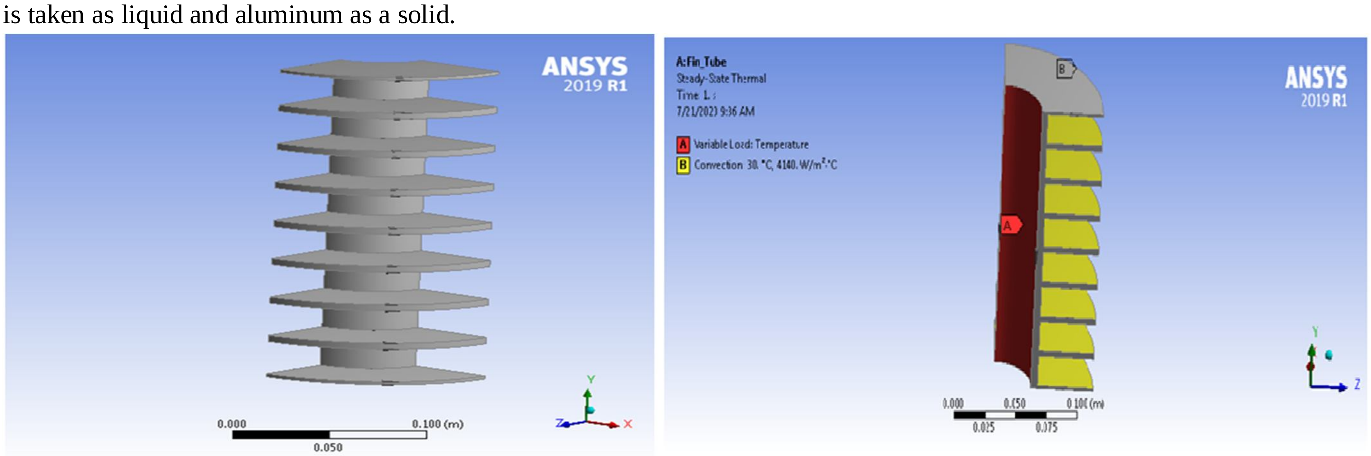

Fig 1.1Geomntry and Boundary Condition

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 8 Issue VIII Aug 2020- Available at www.ijraset.com

V. STEADY STATE WITH RADIATION & CONVECTION HEAT TRANSFER

The same pre-processing steps are followed in the Steady state with radiation & convection heat transfer condition in the ANSYS WORKBENCH. Below Fig 1.2 Shows the design of the fins with tube. The same meshing steps are followed in the Steady state with radiation & convection heat transfer condition in the ANSYS WORKBENCH. Below

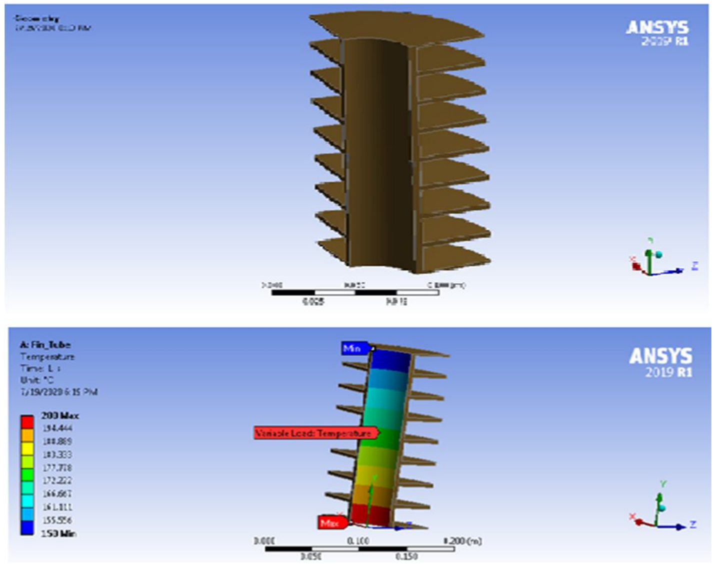

Fig 1.2 Geometry and Boundary Condition

VI. RESULTS AND DISCUSSIONS

Below we discuss the transfer of heat in a steady state with a convection state and a steady state with the radiative heat transfer and convection state obtained from numerical calculations. Temperature and heat flow: Figures appear in Figure 1.3. The maximum temperature is 200 and the heat flow is 3,026e5 and the directional heat flow is 2,808e6 for stable heat transfer with the heat load condition . Temperature and heat flux- Contours are shown in Fig 1.4. The max temp is 200 and heat flux is 3.026e5 and directional heat flux is 2.808e6 for the Steady state with radiation & convection heat transfer condition.

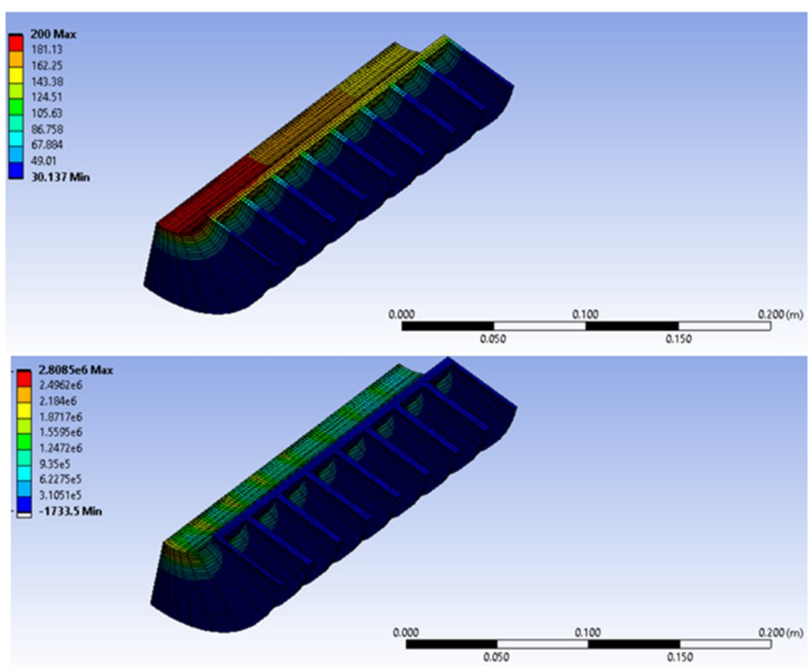

Fig 1.3 Max Temp and directional heat flux

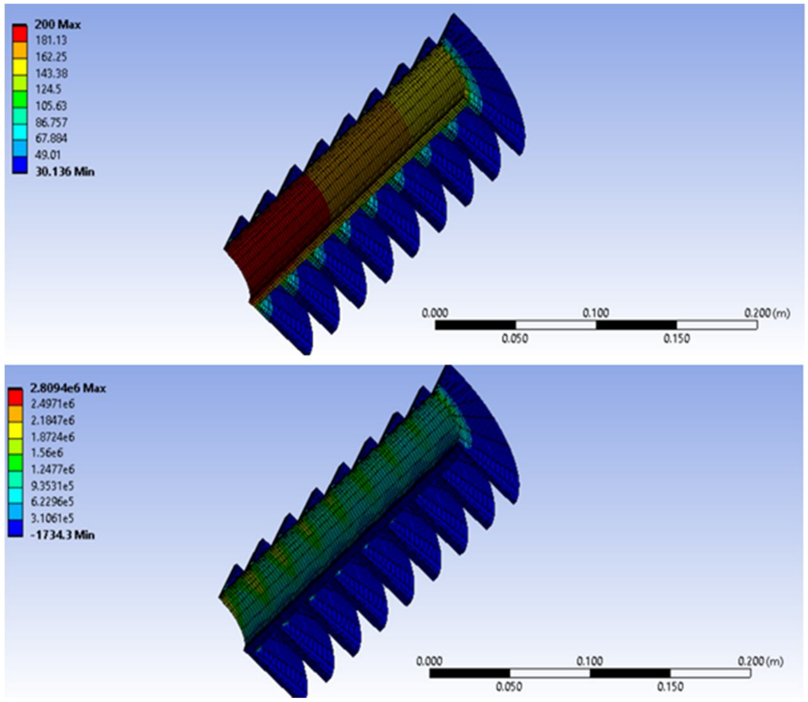

Fig 1.4 Max Temp and directional heat flux

VII. CONCLUSION

3D numerical model with steady state thermal analysis of fin & tube heat exchanger has been analyzed. Temperature load with nonlinear heat transfer coefficient has been applied. In first case we analyzed heat transfer purely with convection cooling. It has been determined the temperature reached max 200 deg C and minimum of 150 deg C along the length of the tube. In second case we considered external fins is subjected to both convection & radiation as well. From energy balance table 5.1 shows it's been observed radiation accounts roughly 2.1 % of heat loss from the system. Major heat loss is due to convection alone.

Table I
Energy Balance

Heat Balance:	Case 1	Case 2
Applied Heat (W)	12125	12135
Convection heat (W)	-12125	-12119
Radiation (W)		-16.356
Net heat Balance (W)	0	0.356

REFERENCES

[1] Kuhn, S., Rudolf von Rohr, P., Experimental investigation of mixed convective flow over a wavy wall. International Journal of Heat and Fluid Flow 29 (2008) 94–106.

[2] Guzman, Amador M., Cardenas, Maria J., Urzua, Felipe A., Araya, Pabolo E., Heat transfer enhancement by flow bifurcations in asymmetric wavy wall channels, International Journal of Heat and Mass Transfer 52 (2009) 3778–3789

[3] Sui, Y., Teo, C.J., Lee, P.S., Chew, Y.T., Shu, C. Fluid flow and heat transfer in wavy microchannels, International Journal of Heat and Mass Transfer 53 (2010) 2760–2772

[4] Sui, Y., Teo, C.J., Lee, P.S., Direct numerical simulation of fluid flow and heat transfer in periodic wavy channels with rectangular cross-sections, International Journal of Heat and Mass Transfer 55(2012) 73-88

[5] Ramgadia, Abhishek G., Saha, Arun K., Fully developed flow and heat transfer characteristics in a wavy passage: Effect of amplitude of waviness and Reynolds number, International Journal of Heat and Mass Transfer 55 (2012) 2494–2509

[6] Naphon, Paisarn, Effect of wavy plate geometry configurations on the temperature and flow distributions, International Journal of Heat and Mass Transfer 36 (2009) 942–946

[7] Mohammed, H.A., Gunnasegaran, P., Shuaib, N.H., Numerical simulation of heat transfer enhancement in wavy microchannel heat sink, International Journal of Heat and Mass Transfer 38 (2011) 63–68

[8] Nandi, Tapas K., Chattopadhyay, Himadri, Numerical investigations of simultaneously developing flow in wavy microchannels under pulsating inlet flow condition, International Journal of Heat and Mass Transfer 47 (2013) 27–31

[9] Kuhn, Simon, Kenjeres, Sasa, Rudolf von Rohr, Philip, Large eddy simulations of wall heat transfer and coherent structures in mixed convection over a wavy wall, International Journal of Heat and Mass Transfer 49 (2010) 1209–1226

[10] Sui, Y., P.S. Lee, Teo, An experimental study of flow friction and heat transfer in wavy microchannels with rectangular cross section International Journal of Heat and Mass Transfer 50 (2011) 2474–2482

[11] Uppada Rama Kanth, Putti Srinivasa Rao, Mallarapu Gopi Krishna, "Mechanical behaviour of fly ash/SiC particles reinforced Al-Zn alloy-based metal matrix composites fabricated by stir casting method", Journal of Materials Research and Technology Volume 8, Issue 1, January–March 2019, Pages 737-744.

[12] U. S. Ramakanth & Putti. Srinivasa Rao, "Wear Behavior of Al 7075/Fa/Sic Hybrid Composites", International Journal of Mechanical and Production Engineering Research and Development (IJMPERD) ISSN(P): 2249-6890; ISSN(E): 2249-8001 Vol. 9, Issue 2, Apr 2019, 331-338.

[13] U. Ramakanth & Putti. Srinivasa Rao, "Corrosion Studies on Ceramic Particulates Reinforced Hybrid Metal Matrix Composites", International Journal of Mechanical Engineering and Technology (IJMET) Volume 10, Issue 01, January 2019, pp. 661–666.