NUMERICAL SIMULATION OF THERMAL BOUNDARY LAYERS IN INDUSTRIAL HEAT EXCHANGERS

Anjaiah Mende
Research Scholar
Arni University
Himachal Pradesh

Dr. Umashanker
Research Supervisor
Arni University
Himachal Pradesh

Dr. Ujenpad Siddeshwar
Research Co-Supervisor
Arni University
Himachal Pradesh

ABSTRACT

Heat transfer rates from a surface can be determined from the slope of the temperature profile measured with a thermocouple wire traversing within a boundary layer. However, accuracy of such measurement can suffer due to flow distortion and conduction through the thermocouple wire. The present numerical study consists of two parts—a 2D simulation of flow distortion due to a cylinder in cross flow near a solid wall and a 3D simulation defined as a fin problem to calculate the thermal profile measurement error due to conduction through the thermocouple wires. Heat exchangers are almost used in every industry. Among them, shell and tube heat exchangers are covering around 32% of the total heat exchanger. Numerical simulation of the Computational models is playing an important role for the prototypes including the Heat Exchanger Models for the improvement in modeling. In this study, the CFD analysis of parallel and counter flow shell and tube heat exchanger was performed. Following project, looked into the several aspects and these are the temperature, velocity, and pressure drop and turbulence kinetic energy along with the heat exchanger length. Hot water was placed in tube side and cold water was placed in shell side of the heat exchanger. Shell side cold temperature was increasing along the heat exchanger length. On the other side, tube side hot water temperature was decreasing along the tube length. This effect was more significance in counter flow rather than the parallel flow. Velocity was more fluctuating in the shell side due to presence of the baffles.

Keywords: *Heat exchanger, Numerical simulation, Boundary layer, Heat Exchanger Models*

INTRODUCTION

Heat exchanger is a device which can transfer heat between two or more fluids from higher temperature to lower temperature. Various types of heat exchangers are used for process of heating and cooling applications. In many engineering sectors heat exchangers are playing major role, due to reliability, technology involved in manufacturing methods, versatility of it and operating at high pressure and temperatures. Since many years, industries like oil refinery, power generation plants are using shell and tube heat exchangers, in it baffles plays the key component to leads the fluid flow in shell area. The characteristics of the fluid flow and the temperature profile of the fluid once the wall temperature changes, is completely dependent on the fluid properties due to wall temperature. The mass and energy transfer within the velocity and thermal boundary layer largely govern near-wall heat loss. In engines, these boundary layers are unsteady; the outer fluid is not constant in velocity, temperature, or pressure. This unsteady behavior deviates from boundary layer theory and is not fully understood. Most wall functions in heat transfer models do not account for such deviations. Consequently, they struggle to predict engine heat loss accurately. This is troubling for engine simulations employing such models because they become a less effective design tool.. The only assumption here is to make sure that both the temperature and the pressure profiles are starting at the same point. The velocity field of the fluid that is flowing through the wall temperature tubing was described based on The benefits of using an extended surface for heat transfer have received considerable attention, particularly for gas flow, as the PFHEs are mainly employed for liquid-gas and gas-gas applications. By using fins configured for extended surfaces, we can enhance the heat transfer coefficients, however the pressure-drop penalties are high. The friction factor is increased where discontinuous type of fin geometry was adopted to interrupt the boundary layers. Friction drag is the friction of a fluid against the surface that it is moving through. It is directly proportional to the area of the surface in contact with the fluid and increases with the square of the velocity. Although friction drag is associated with the high heat transfer coefficient, form drag has no counterpart and represents wasted energy. The form drag can be substantial depending on the quality of the cutting edge. However, machined-formed fins are generally free from this problem.

Heat transfer from a hot or cold object exposed to an external fluid flow is arguably the most basic form of forced convection heat transfer encountered in industrial processes and technological applications. When analyzing this category of heat transfer problems, in many cases, it is well justified to assume that the fluid properties such as density ρ, viscosity μ, specific heat cp, and thermal conductivity k are constant. In general, this assumption renders the energy equation (the partial differential equation that governs the distribution of the fluid temperature) linear and decoupled from the Navier-Stokes equations (from which the fluid velocity field is determined). There exist, however, practical cases where at least one of the fluid properties cannot realistically be considered constant. For instance, it has been shown that the effective thermal conductivity of nano fluids increases considerably with the rising temperature or, for liquid metals, k has been found to vary roughly linearly with the temperature in a wide range of operating conditions.

LITERATURE REVIEW

Konfe, A,. and Kam, S. (2024), In this paper, we propose a thermal model of a hybrid photovoltaic/thermal concentration system. Starting from the thermal balance of the model, the equation is solved and simulated with a MATLAB code, considering air as the cooling fluid. This enabled us to evaluate some of the parameters influencing the electrical and thermal performance of this device. The results showed that the temperature, thermal efficiency and electrical efficiency delivered depend on the air mass flow rate. The electrical and thermal efficiencies for different values of air mass flow are encouraging, and demonstrate the benefits of cooling photovoltaic cells. The thermal efficiency of the solar cell increases as the light concentration increases, whatever the air flow rate used. An interesting and useful finding was that the proposed numerical model allows the determination of the electrical as well as thermal efficiency of the hybrid CPV/T with air flow as cooling fluid.

Nogueira, É. (2023), This work contributes to the improvement of energy-saving in air conditioning systems. The objective is to apply the thermal efficiency of heat exchangers for localized determination of the thermal performance of heat exchangers with individually finned heat pipes. The fundamental parameters used for performance analysis were the number of fins per heat pipe, the number of heat pipes, the inlet temperatures, and the flow rates of hot and cold fluids. The heat exchanger under analysis uses Freon 404A as a working fluid in an air conditioning system for cooling in the Evaporator and energy recovery in the Condenser. The theoretical model is localized and applied individually to the Evaporator, Condenser, and heat exchanger regions. The results obtained through the simulation are compared with experimental results that use a global approach for the heat exchanger. The thermal quantities obtained through the theoretical model in the mentioned regions are air velocity, Nusselt number, thermal effectiveness, heat transfer rate, and outlet temperature.

Inyang, U. and Uwa, I. (2022), The research paper tends to review the effectiveness of helical coil in heat exchangers (HCHE). Heat exchanger is a device used in transferring thermal energy between two or more fluids or solid interfaces and a fluid, in solid particulates and a fluid at different temperatures and thermal contact. The author has concisely discussed the helical coil in heat exchanger at different shapes and conditions and compared the HCHE with straight tubes heat exchangers, and the factors affecting the performance and effectiveness of the helical coil heat exchanger such as the curvature ratio, and other heat exchangers. The author demonstrated that the HCHE provided more excellent heat transfer performance and effectiveness than straight tubes and other heat exchangers because of secondary flow development inside the helical tube, and heat transfer coefficient increased with an increase in the curvature ratio of HCHE for the same flow rates. The secondary flow and mass flow rates, advantages and disadvantages have also been reviewed.

S. Harisha (2021), Numerical methods are described as techniques by which several mathematical problems are formulated, because they may be easily solved with arithmetic operations. These methodologies have a great impact on the current development of finite element theory and other areas. We have given a short study of numerical methodologies applied in fluid flow and heat and mass transfer problems in mechanical engineering which includes finite difference method, Finite element method, Boundary value problems (general), Lattice Boltzmann's methods, Crank-Nicolsan scheme methods, boundary integral method, Runge-Kutta method, Taylor series method and so on. We have discussed some phenomena taking place in fluids such as surface tension, coning, water scattering, Stokes law, gravity-capillary, and unsteady free surface flows, swirling, and so on. We have also analyzed boundary value problems on boundary problems, eigenvalue problems and found a numerical way to solve these problems.

Heat Exchanger

A Heat exchanger is a piece of equipment built for efficient heat transfer from one medium to another. The media may be separated by a solid wall, so that it never mix or may be in direct contact. The heat exchangers widely used in space heating, refrigeration, air conditioning, power plants, chemical plants, petrochemical plants, petroleum refineries, and natural gas processing and sewage treatment. One common example of a heat exchanger is the radiator in a car , in which the heat source , being a hot engine-cooling fluid, water transfer heat to air flowing through the radiator. The plate heat exchanger is widely recognized today as the most economical and efficient type of heat exchanger on the market. With its low cost, flexibility, easy maintenance and high thermal efficiency, it is unmatched by any other type of heat exchanger. The corrugation patterns that induce turbulent flows, it not only achieves unmatched efficiency but also creates a self-cleaning effect thereby reducing fouling. The most common surface pattern used is the chevron design.

Compact Heat Exchanger

In forced-convection heat transfer between a gas and a liquid, the heat transfer coefficient of the gas may be 10 to 50 times smaller than that of the liquid. The use of specially-configured surfaces can be used to reduce the gas side thermal resistance. For heat transfer between two gases, the difficulty in inducing the desired heat exchange is even more pronounced. In this case especially, the use of enhanced surfaces can substantially reduce heat exchanger size. This is the motivation behind the design of a category of heat exchangers with reduced size and greatly enhanced gas side heat transfer, which are referred to as "compact A plate fin heat exchanger is a type of heat exchanger that uses metal plates to transfer heat between two fluids. This has a major advantage over a conventional heat exchanger in that the fluids are exposed to a much larger surface area because the fluids spread out over the plates. This facilitates the transfer of heat, and greatly increases the speed of the temperature change.

Printed Circuit Heat Exchanger

Heatric is a wholly owed subsidiary of Meggitt PLC and is a manufacturer of Printed Circuit Heat Exchangers for gas processing and petrochemical markets. PCHEs are constructed from flat metal plates into which fluid flow channels are chemically milled. The chemical milling technique is similar to that employed for etching electrical printed circuits, and hence is capable of producing fluid circuits of unlimited variety and complexity. The channel may be straight or wavy. The milled plates are then stacked and diffusion bonded together to form strong, compact, all-metal heat exchange cores (see figure 3). The diffusion bonding is a "solid-state joining" process entailing pressing metal surfaces together at temperatures below the melting point, thereby promoting grain growth between the surfaces. The blocks are then welded together to form the complete heat exchange core for the thermal duty. Finally, fluid headers and nozzles are welded to the core in order to direct the fluids to the appropriate sets of passages.

Heat and Mass transfer

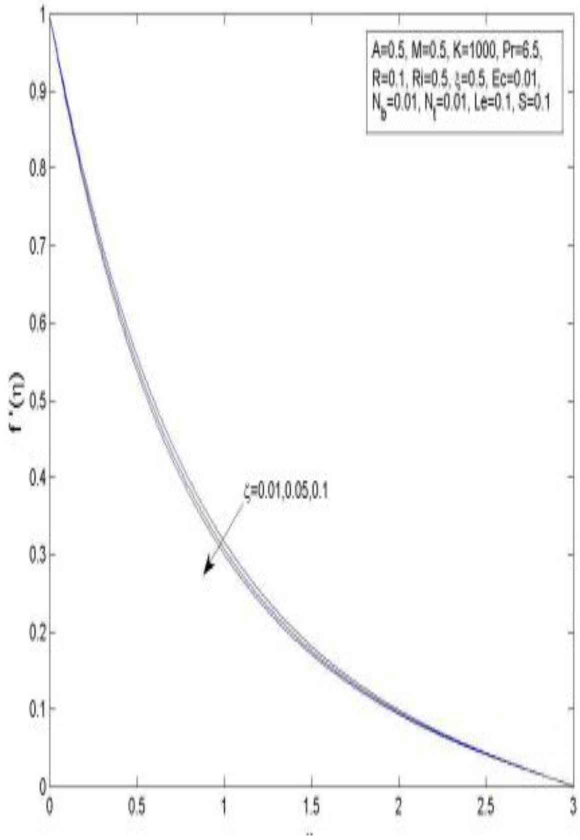
Antonopoulos proposed a solution for problems in heat conduction with their boundary conditions on nonhomogeneous composite regions like slabs, cylinders, etc., using the separation of variables method and orthogonal expansion methods for both the surfaces. Although authors have tried to answer the heat conduction problem using boundary value methods in a multilayer structure and Tittle has solved a heat conduction problem using Laplace transform methods. Hence, the author solves the outer boundary problem for the regional layers on both the boundaries (by the outer boundary surface and the surface fluids) by acknowledging the problem as a non-homogeneous problem and solving it by the orthogonal expansion and method of separation of variables.

METHODOLOGY

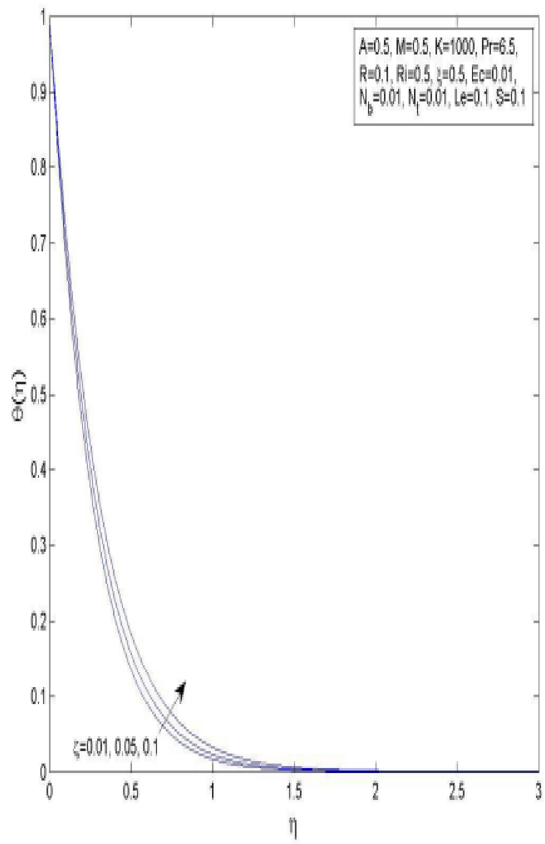
The work which is undertaken for the thesis is to evaluate the performance of longitudinal vortex generator, which is used to enhance the transfer rate in a plate fin-heat exchanger. These vortex generators are mounted inside the triangular fins which are used as inserts between the plates. For detailed investigation a rectangular winglet pair is considered. In manufacturing processes quality of the final product usually depends on the rate of heat transfer at the stretching surface. Dissipative heat is the heat generated due to internal friction with electric current flows through resistance it converted into heat, this effect is called Joule heating. Also, the thermal radiation effect plays a significant role in manufacturing and designing of reliable equipment, gas turbines, nuclear plants, various propulsion devices for aircraft, satellite and space vehicles. It controls the heat transfer process in polymer processing. To manage the heat, transfer characteristic, porous media adjacent to the surface can be used. The problems related to stretching sheet occurred in polymer industry were noticed and analyzed. Heat and mass transfer on stretching sheet with suction or blowing was investigated studied the stretching problem for a surface moving with linear velocity and variable temperature. Considered heat transfer on a continuous stretching surface with suction and blowing.

RESULTS AND DISCUSSIONS

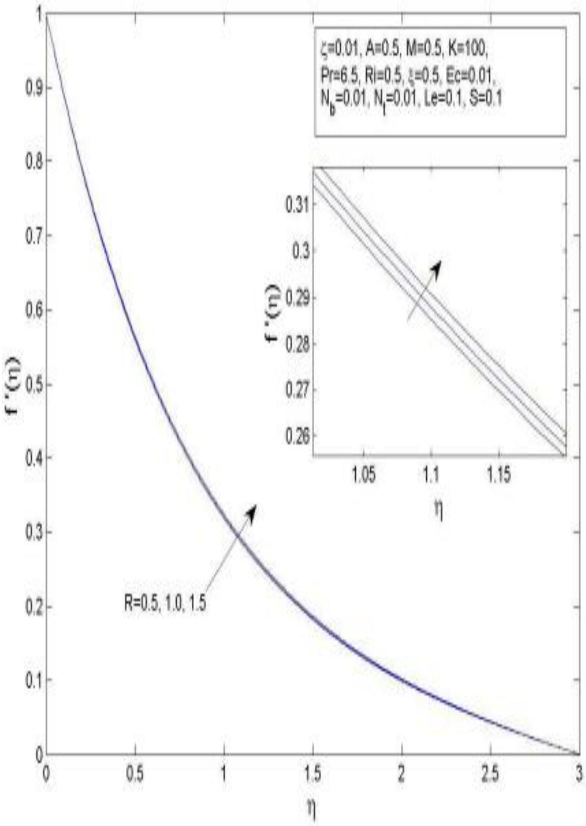
Graphs 1 and 2 depict that due to increase in solid volume fraction of nano fluid, fluid velocity decreases while temperature increases. The physics behind this is velocity boundary layer thickness decreases and thermal conductivity enhances due to increase in nano particle volume fraction. It is noted from Graphs 3 and 4 that fluid velocity and fluid temperature increase by uplifting radiation parameter, as fluid particles consume the radiated heat from the heated surface. Enhancement in Prandtl number results in fall of fluid velocity and fluid temperature, due to which skin friction decreases.



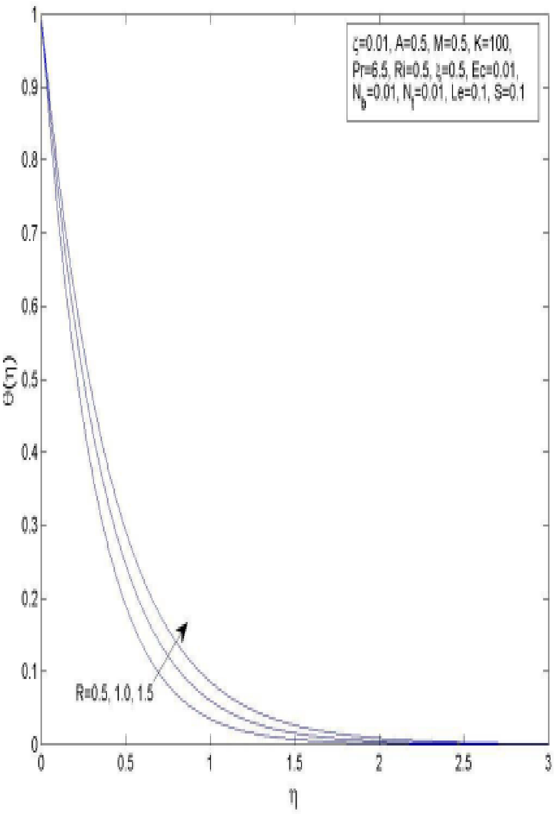
Graph 1 Velocity profiles versus η for different values of ζ .



Graph 2 Temperature profiles η versus for different values of ζ



Graph 3 Velocity profiles versus η for different values of R.



Graph 4 Temperature profiles versus η for different values of R.

CONCLUSION

Moreover, the study revealed that boundary layer development and flow regime transitions play a crucial role in determining the temperature gradients and heat flux distributions. It was observed that increasing the Reynolds number significantly alters the velocity field, leading to enhanced convective heat transfer. Similarly, variations in the Prandtl number were found to influence the thermal boundary layer thickness, impacting the overall heat transfer coefficient. These dependencies underscore the importance of dimensionless numbers in characterizing and scaling fluid flow and heat transfer processes. Numerical simulations using finite difference and finite volume methods also provided key insights into the limitations and advantages of each numerical scheme. While the finite difference method was computationally efficient for simple geometries, the finite volume method offered more flexibility and accuracy for complex domains with irregular boundaries. Grid independence studies and convergence analysis were carried out to ensure the reliability of results, further strengthening the confidence in the model predictions. Another essential contribution of the study was the identification of critical zones within the domain where thermal stresses and flow separations are likely to occur. These findings have practical engineering implications, particularly in the design and optimization of heat exchangers, pipelines, and microfluidic devices. By identifying areas of high thermal gradients, engineers can proactively modify designs to improve efficiency and prevent material fatigue or failure due to thermal expansion and contraction cycles.

REFERENCES

1. *Konfe, A,. and Kam, S. (2024), "Numerical Assessment of the Thermal Efficiency of a Concentrated Photovoltaic/Thermal (CPV/T) Hybrid System Using Air as Heat Transfer Fluid." Smart Grid and Renewable Energy, Vol.15 No.1, 1-14 ISSN Online: 2151-4844.*
2. *Nogueira, É. (2023), "Localized Theoretical Analysis of Thermal Performance of Individually Finned Heat Pipe Heat Exchanger for Air Conditioning with Freon R404A as Working Fluid." Journal of Materials Science and Chemical Engineering, Vol. 11 No.8, 61-85 ISSN: 2327-6053.*
3. *Inyang, U. and Uwa, I. (2022), "Heat Transfer in Helical Coil Heat Exchanger." Advances in Chemical Engineering and Science, Vol.12 No.1, 26-39 ISSN: 2160-0406.*
4. *S. Harisha (2021), "A Study On Numerical Methodologies In Solving Fluid Flow And Heat Transfer Problems." Earth and Environmental Science. ISSN 2634-8845,doi:10.1088/1755-1315/850/1/012021.*
5. *Rao, A,and Beg, O. (2015), "Numerical Modeling of Non-Similar Mixed Convection Heat Transfer over a Stretching Surface with Slip Conditions." World Journal of Mechanics, Vol.5 No.6, 117-128 ISSN: 2160-0503.*
6. *Men, Q,and Meng, X. (2014), "Calculation Method of Passive Residual Heat Removal Heat Exchanger and Numerical Simulation." Journal of Power and Energy Engineering, Vol.2 No.9, 8-1ISSN: 2327-59014.*
7. *Hasan, M,and Yaghoubi, M. (2012), "Investigation of a Counter Flow Micro channel Heat Exchanger Performance with Using Nano fluid as a Coolant." Journal of Electronics Cooling and Thermal Control, Vol.2 No.3, 35-43 ISSN: 2162-6170.*
8. *Gireesha, B,and Bagewadi, C. (2011), "Boundary Layer Flow of an Unsteady Dusty Fluid and Heat Transfer Over a Stretching Sheet with Non-Uniform Heat Source/Sink." Engineering, Vol.3 No.7, 726-735 ISSN: 1947-394X.*
9. *Abbassi, F,and Shahmardan, M. (2017), "Numerical Study of Heat Transfer and Flow Bifurcation of CuO Nano fluid in Sudden Expansion Micro channel Using Two-Phase Model." Modern Mechanical Engineering, Vol.7 No.2, 57-72 ISSN: 2164-0181.*
10. *Anil Singh Yadav (2013), "Heat transfer and fluid flow analysis of solar air heater: A review of CFD approach." Renewable and Sustainable Energy Reviews. Volume23, Pages.60-79, ISSN:1364-0321,https://doi.org/ 10.1016/ j.rser.201 3 .02.035.*