LITERATURE REVIEW

A cooling system is used to remove this excess heat. Most automotive cooling systems consist of the following components: radiator, water pump, electric cooling fan, radiator pressure cap, and thermostat. Of these components, the radiator is the most prominent part of the system because it transfers heat [1].

![Components within an automotive cooling system](image)

**Figure 2.1 Components within an automotive cooling system**

As coolant travels through the engine's cylinder block, it accumulates heat. Once the coolant temperature increases above a certain threshold value, the vehicle's thermostat triggers a valve which forces the coolant to flow through the radiator.

As the coolant flows through the tubes of the radiator, heat is transferred through the fins and tube walls to the air by conduction and convection.
2.2 Radiator & Tubes Construction:

Figure 2.2 Radiator & tubes Construction.

Figure 2.3 Anatomy of Fin & Tube Heat Exchanger
2.3 Types of Radiator:

Type & shape of tubes:

Figure 2.4 Radiator
Factors affect Radiator Performance:

- Air Turbulence - increase air turbulence, thereby increasing convection
- Air Speed - Increase the speed of the air flowing over the radiator
- Radiator Tubes - Make the tubes smaller and increase the total amount of tubes - This concept decreases the time it takes to transfer the same amount of heat by exposing it in more places within the radiator.
- Surface Area (Fins) - Increase the surface area of the fins,
- Surface Area (Total) Increase the overall surface area of the radiator.
- Coolant - Anti oxidant.
Figure 2.6 Enhancement of heat transfer
Existing Rectangular Radiator:

Figure 2.7 Rectangular Radiator
Square shaped heat exchanger:

Fig 2.8 shows a square-shaped heat exchanger with a fan provided to deliver air in a circular area.

If the length and breadth of the heat exchanger is equal to D, the effective area of such heat exchanger will be equal to $D^2$.

While the flow of air from the fan (without shroud) will be of area $(\pi/4) \ D^2 = 0.76 \ D^2$.

The difference in the area of the square and the circle would be $(D^2 - (\pi/4) \ D^2) = 0.24 \ D^2$. 
Velocity profile of an air-cooled heat exchanger using a fan:

- Velocity of air generated by the fan is not constant along its axial direction.
- It is found to be almost zero at the centre and gradually increases at the rate of square of the radius.

Figure 2.9 Velocity profile of an air-cooled heat exchanger
Proposed Improved Heat Exchanger:

In the existing design, there is no heat transfer area at the centre where the flow of air is almost zero.

In the proposed design the tubes and the fins are so arranged that the outlet air has nearly constant velocity.

**Fins:** They are the most important component of any heat exchanger. They are used to transfer the heat generated inside the heat exchanger to the surroundings reducing inside temperature.
Figure 2.11 Fin Geometry of Proposed Radiator

- Smaller fins are provided at the centre and longer fins at the periphery.
- The length of fins are so adjusted that the velocity of air coming out of the heat exchanger remains constant over the effective area.

Figure 2.12 Low Velocity Zones

- It can be observed that low velocity or stagnation zones are created in the corners hence may be eliminated and circular radiator can be developed for optimum efficiency.

Due to its variety of advantages offered, the progress of the studies on micro devices used in chemical analysis is observed. The micro heat exchangers are well recognized for their higher performance. The applications of them are ranging from increasing of heat transfer applications to chemical reactions or evaporation of liquids applications. The current paper is addressing an engineering approach for modelling the heat and mass transfer processes in micro heat exchangers. The approach is based on the dimensional analysis and principles of theory of similitude that allow the modelling of microscale systems using a physical system at miniscale[1].
Heat Exchanger performance in an automotive HVAC is critically influenced by the uniformity of flow through the core face on the air side. Performance is generally measured in terms of both Core Duty and Pressure Drop. HVAC units are typically tightly constrained by packaging requirements imposed on components under the Instrument Panel resulting in misdistribution of flow across typical cross-sections. In order to improve heat exchanger performance, such misdistribution upstream of the device must be rectified without infringement of packaging boundaries. In this paper, we show how vanes, or baffles, may be used in automotive HVAC units to achieve certain objectives relating to flow uniformity upstream of heat exchangers. CFD is used as a means of capturing the impact of design changes on performance objectives. A "guide vane" with qualitatively desirable characteristics is parameterized, and the ANSA morphing capability used to introduce the parameterized feature into the flow domain. The parametric representation permits examination of the design space in search of optimum solutions with respect to the objectives. Formal optimization methods are used to drive the solution search. ANSA™ is used for morphing and meshing, FLUENT® is used to perform CFD, and iSIGHT-FD™ is used for all optimization algorithm[2].

Space-radiator materials and fin-tube geometry have a large influence on the ultimate size and weight of the radiator. Many factors, in turn, influence the requirements for materials and geometry, such as meteoroid damage protection, structural integrity, vehicle integration, and fluid compatibility. Similarly, there is a wide variety of materials and geometries that can be considered. It appears that a completely satisfactory solution of the materials-geometry question for advanced Rankine power systems is not yet in hand. Designers of advanced nuclear space power systems are very much concerned with the size and weight of the radiators required to dissipate the waste-heat loads. The radiator size' and weight are each governed by several principal factors or variables: The size of the radiator is determined by the 'design heat load and the external radiation heat-transfer rate. External radiation heat transfer is a function primarily of the temperature of the working fluid, the emittance of the radiator surface, the specific radiator geometry and material, and the internal fluid flow. The temperature of the radiator working fluid is determined by the cycle optimization and design, which has been the subject of many system studies. The surface emittance, for many radiator materials, will depend on the applied coating[5].
This paper discusses two spacecraft radiators currently under study by the NASA Lewis Research Center: the Liquid Droplet Radiator and the Liquid Belt Radiator. These advanced concepts offer benefits in reduced mass, compact stowage, and ease of deployment. Operation and components of the radiators are described, heat transfer characteristics are discussed, and critical technologies are identified. Finally, the impact of the radiators on large power systems is assessed. With the advent of the Space Station, there is increased interest in large power systems for space. When current technology is scaled to Space Station initial operating condition power levels, the radiator comprises 30 to 50 percent of the total power system mass. Therefore, the radiator is an obvious target for reducing system mass, and thus launch cost[6].

The technical requirements of a shuttle-attached Liquid Droplet Radiator (LDR) experiment are discussed. The Liquid Droplet Radiator is an advanced lightweight heat rejection concept that can be used to reject heat from future high powered space platforms. In the LDR concept, sub millimeter sized droplets are generated, pass through space, radiate heat before they are collected and recirculated back to the heat source. The LDR experiment is designed to be attached to the shuttle longeron and integrated into the shuttle bay using standard shuttle/experiment interfaces. Overall power, weight, and data requirements of the experiment are detailed. Shuttle integration and safety design issues are discussed. An overview of the conceptual design of the experiment is presented. Future space applications will require the development of advanced lightweight heat rejection systems. The liquid droplet radiator is a heat rejection system which holds much promise for fulfilling the need for lightweight heat rejection. To establish feasibility in a relevant environment, the liquid droplet radiator is proposed for a space flight test aboard the space shuttle. This document will detail the development of the droplet radiator, discuss technical issues which the proposed experiment will address, and detail how the experiment will be integrated into the shuttle bay[8].

The paper deals with the application of Parallel Evolutionary Algorithms (PEA) and the Finite Element Method (FEM) in shape optimization of heat radiators. The fitness function is computed with the use of the coupled thermoelasticity modeled by MARC/MENTAT software. The geometry, mesh and boundary conditions are created on the basis of a script language implemented in MENTAT. In order to reduce the number of design parameters in evolutionary algorithms, the shape of the structure is modeled by Bezier curves. Numerical examples for some
shape optimization are included. In the present paper, the application of Parallel Evolutionary Algorithms (PEA) and commercial FEM software MARC/MENTAT for shape optimization of heat radiators are presented. Evolutionary algorithms have had various applications to structural optimization. The main feature of this class of procedures is their randomness[10].

A low-power consumption, small-size smart antenna, named electronically steerable parasitic array radiator (ESPAR), has been designed. Beamforming is achieved by tuning the load reactances at parasitic elements surrounding the active central element. A fast beam forming algorithm based on simultaneous perturbation stochastic approximation with a maximum cross correlation coefficient criterion is proposed. The simulation and experimental results validate the algorithm. In an environment where the signal-to-interference-ratio is 0 dB, the algorithm converges within 50 iterations and achieves an output signal-to interference- plus-noise-ratio of 10 dB. With the fast beam forming ability and its low-power consumption attribute, the ESPAR antenna makes the mass deployment of smart antenna technologies practical[11].

For many industrial processes, the chimney is the final step before hot fumes, with high thermal energy content, are discharged into the atmosphere. Tapping into this energy and utilizing it for heating or cooling applications, could improve sustainability, efficiency and/or reduce operational costs. Alternatively, an unused chimney, like the monumental chimney at the Eindhoven University of Technology, could serve as an “energy channeler” once more; it can enhance free cooling by exploiting the stack effect. This study aims to identify design parameters that influence annual heat exchange in such stack chimney applications and optimize these parameters for specific scenarios to maximize the performance. Performance is defined by annual heat exchange, system efficiency and costs. The energy required for the water pump as compared to the energy exchanged, defines the system efficiency, which is expressed in an efficiency coefficient (EC). This study is an example of applying building performance simulation (BPS) tools for decision support in the early phase of the design process. In this study, BPS tools are used to provide design guidance, performance evaluation and optimization. A general method for optimization of simulation models will be studied, and applied in two case studies with different applications (heating/cooling), namely[15].

In this study, a system model is developed for a space cooling system with a focus on the finned-tube condenser design details using a new environmentally friendly refrigerant as the
working fluid. An optimization algorithm is implemented to find an optimum design for 10 condenser design parameters using various constraints. The figure of merit was system efficiency. Though it is not possible to prove that the search method will give the best global design available, it did find a significantly better design than current practice. The optimum condenser design was found to give the same performance as a coil optimized through a manual search costing 23% more. It is also shown that the optimum design is consistent with minimum entropy generation for both the condenser component and the total system. This design optimization methodology is fully developed and presented in the paper so that it can be applied to other energy system’s heat exchanger optimization opportunities[16].

The concentric tube heat exchanger was designed in order to study the process of heat transfer between two fluids through a solid partition. It was designed for a counter-flow arrangement and the logarithmic mean temperature difference (LMTD) method of analysis was adopted. Water was used as fluid for the experiment. The temperatures of the hot and cold water supplied to the equipment were 87°C and 27°C, respectively and the outlet temperature of the water after the experiment was 73°C for hot and 37°C for cold water. The results of the experiment were tabulated and a graph of the mean temperatures was drawn. The heat exchanger was 73.4% efficient and has an overall coefficient of heat transfer of 711W/m²K and 48°C Log Mean Temperature Difference. The research takes into account different types of heat exchangers[17].

The objective of this paper is to analyze through the Finite Elements Method (FEM) and to dimensional optimize the frontal radiators of cooling system for electric power transformer by type TTU 630 kVA 20/0.4 kV in terms of maximum heat transfer. Finite Elements Analysis was performed using SolidWorks 3D CAD Design and COSMOS Flow Works 2008 Software. The proposed method results in the economical design of the electric power transformer by type 630 kVA 20/0.4 kV significantly reduce the cost of manufacturing transformer[18].

This paper is intended to assist anyone with some general technical experience, but perhaps limited specific knowledge of heat transfer equipment. A characteristic of heat exchanger design is the procedure of specifying a design, heat transfer area and pressure drops and checking whether the assumed design satisfies all requirements or not. The purpose of this paper is how to design the oil cooler (heat exchanger) especially for shell-and-tube heat exchanger which is the majority type of liquid-to-liquid heat exchanger. General design
considerations and design procedure are also illustrated in this paper and a flow diagram is provided as an aid of design procedure. In design calculation, the MatLAB and AutoCAD software are used. Fundamental heat transfer concepts and complex relationships involved in such exchanger are also presented in this paper. The primary aim of this design is to obtain a high heat transfer rate without exceeding the allowable pressure drop. This computer program is highly useful to design the shell-and-tube type heat exchanger and to modify existing design[20].