ISSN (Online) : 2454-4159
International Journal of Advanced Research in Science Management and Technology

Design of Liquid Cooling Premises Using CFD for High Heat Dissipation Electronic Boards

Pankaj Jivan Patel[1], Kailas Tukaram Patil[2], Mankalal Hiraji Patil[3]

PG Scholar, Dept.of Mechanical Engg., P.S.G.V.P. Mandal's D.N.Patel College of Engg., Shahada, M.S., India[1]
Associate Professor, Dept.of Mechanical Engg., P.S.G.V.P. Mandal's D.N.Patel College of Engg., Shahada, M.S., India[1,2]

ABSTRACT- In datacenter, pharmaceutical industry and many other places minimum or low temperature required. Now days higher power electronic systems is used it large amount of heat dissipate but for such electronic systems cooling technology not much more change. Because of large heat dissipation few times electronic device like laptop burns. Also some time thermal injuries placed in a patient's. From many year study conducted for fluid flow and heat transfer behavior in microchannel heat sinks for application in electronic cooling. Cooling generally two types i.e. air cooling and liquid cooling. Liquid cooling compare to air cooling is more effective for cooling on electronic chip (electronic device).

The CFD (Computational Fluid Dynamics)) IS powerful engineering analysis tool. The use of CFD to enable the predictions of the temperature distribution in circuit boards i.e. electronic chips The CFD model was required to incorporate the effects of mixed convection on surfaces, heat generation in the modules, and conduction inside the board. Appropriate convection heat transfer coefficients and boundary conditions resulted in a temperature distribution in the board and chips.

KEYWORDS: liquid cooling, heat dissipation rate, electronic chips, CFD simulation

I. INTRODUCTION

Today's life is very modern the use of electronic equipment is very practically from toys and appliance to high power computers. In the electronic components depends on passage of electric current to perform their work but in this work they excessive heat mean dissipate large amount of heat (heat generation due to the current flow through a resistance). High rate of heat generation and its operating temperature is directly proportional. The failure of electronic equipment increases exponentially with temperature. Another problem is high thermal stresses In electronic components mounted on circuit boards result is temperature variations is one of the reason to failure electronic devices.[1] Therefore, thermal control and cooling of system is important in the design and operation of electronic equipment. The major causes of electronic failure as shown in table 1.1[2]

Table 1. Major causes of electronic failure

Causes of electronic Failure	percentage
Temperature	55
Vibration	20
Humidity	19
Dust	6

Now discuss with type of cooling method. Mainly air cooling and liquid cooling systems consider.

1) **Air cooling**

For thermal control Air cooling is the simplest and principal method and it most widely used for variety of electronic systems ranging from small electronics devices/system to large business systems. The advantages of air cooling are its ready availability and ease of application. At the end 1964, nearby all IBM computers were cooled by force air. In many cases air moving devices are installed at the bottom or top of a column of boards to provide sufficient cooling. For low power electronic devices or low capacity or low heat dissipation devices air cooling is used is effectively. Air cooling is two types natural and forced convection. When natural convection is not sufficient then forced convection is used by a fan or blower. For high power electronic high temperature mean high heat flux it difficult to maintain the temperature in range. This is one of the main reasons why the air cooling option becomes unattractive, and eventually not viable for very high heat flux conditions. [2]

2) **Liquid cooling**

Now a days for higher power dissipation and more heat sources continues, a more attractive solution is to use closed liquid cooling loops to efficiently spread heat to finned surfaces and it situated anywhere within these electronics system.

It is important that if uses of liquid cooling mean air cooling is eliminated because liquid cooling is efficient heat spreaders internally only and within the electronics system but the electronic system is still cooled by air from external perspective.

At the time of cooling system design the type of liquid, fluid and system pressure, cold plate weight, type of material, fluid flow, inlet temperature, and the allow-able or desired pressure drop are important to consider. The type of liquid is various but optimum cooling choice is plane water it used only in controlled environments, laboratory conditions, or requested solutions. The use of tap water attack inside of aluminum flow channel because it contain active ions or other impurities. May be after some time aluminum channels will corrode, causing a leak path and ultimately equipment failure. That is why copper in tube or channel form is the preferred solution with water. [3]

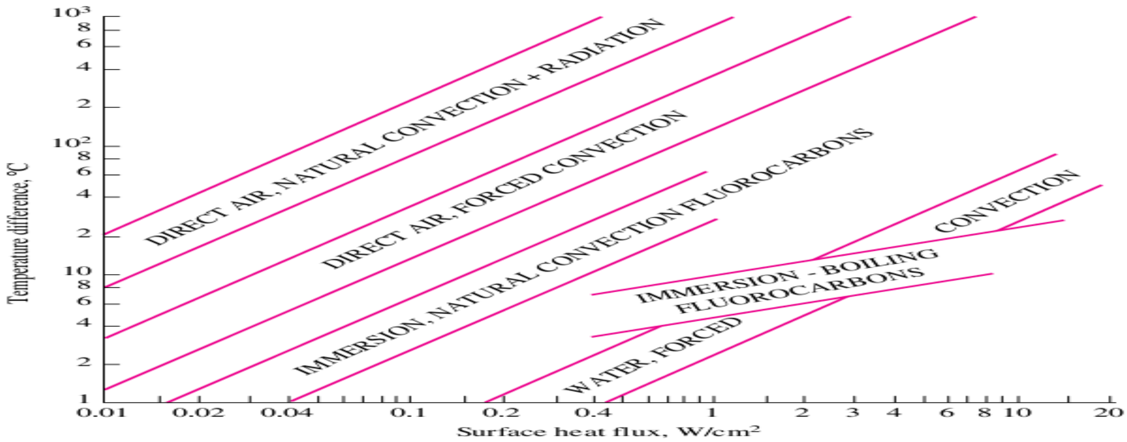


Fig1: Heat fluxes that can be attained at specified temperature differences with various heat transfer mechanisms and fluids. [1]

The fluid flow channels and substrate can be arranged in different way with different configurations depending on device size and power dissipation requirement. The type of cold plate configurations are as follow,

1) Formed Tube Cold Plate

The coolant tubes are joined to the cold plate substrate by using a thermal epoxy. In this design, copper plate is generally used, but sometimes aluminum plate is also used for low power applications. This is one of the simplest cold plate designs, but its performance is rather poor, limiting its use in the low-power applications. [3]

2) Deep Drilled Cold Plate

As the power dissipation increases, the contact resistance of the plate and the tube wall become unacceptably high. In this design, deep holes are drilled in the plane of the substrate plate, generally made of copper. These holes are then configured with end caps to create coolant flow paths through the substrate. The placement of the electronic devices often influences the coolant passage layout. [3]

3) Machined Channel Cold Plates

If heat flux and power increase then necessary to improve the thermal performance of the channels. In this design, channels are machine-cut into the base plate and a cover is soldered in place to form the flow passages. Depending on the thermal performance desired, these channels can be several millimeters wide or as small as 200µm wide microchannels for extremely high heat flux applications. [3]

4) Pocketed Folded-Fin Cold Plates

The local heat transfer coefficient, as well as the surface area in the coolant passages, can be enhanced by implementing fins in the coolant passages. In this design, recessed pockets are machined to accept various folded fin inserts, which are soldered inside the passages. [3]

Two phase cooling system

The refrigerant-based, two phase cooling system compare with water cooling it gives double the thermal performance with only 25% of flow rate required. This allows for smaller pumps and fluid lines, which results in less power required to cool the equivalent system whether a battery, motor, power module, or combination of all three. [4]

The refrigerant used in two-phase evaporative cooling (closed loop system it working shown in fig.) system is non-conductive and non-corrosive for cooling electronic system. It gives, highly efficient system that is fully modular, as well as lighter, safer, and more reliable than traditional thermal systems. [6]

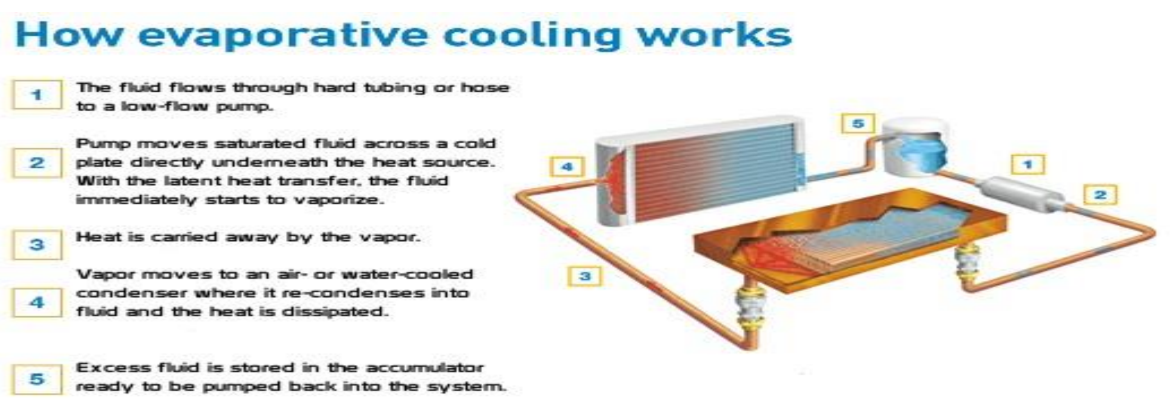


Fig 2: working procedure of two phase cooling system [6]

The System uses a small pump to deliver just enough coolant to the evaporator - usually a series of one or more cold plates optimized to acquire the heat from the device being cooled. The two-phase coolant vaporizes to maintain a cool, uniform temperature on the surface of the device. Then, the vaporized coolant is pumped to a heat exchanger, where it rejects heat to the ambient, and condenses back into a liquid, completing the cycle. [4] [5]

Heat transfer coefficient

Heat transfer coefficient is for various cooling method is shown in table.

Where, h= heat transfer coefficient,

ΔT= temperature difference.

Q = heat transfer per unit time

A= surface area

	h (w/m^2k)	Q (w)
Air natural convection	5-30	0.02-0.09
Air forced convection	20-400	0.06-1.2
Liquid forced convection	100-1600	0.3-4.8
Liquid immersion boiling	800-10000	2.4-30
Direct spray	1000-30000	6-90
Assumption $Q=h*A*\Delta T$, $A=1*1$ cm^2, $\Delta T=30$ ^0C		

II. CFD

Computational Fluid Dynamics is the branch of fluid dynamics providing a cost effective means of simulating real flows by the numerical solution of the governing equation.

Computational fluid dynamics (CFD), the field of study of solution of the equations of fluid flows through use of a computer. Modern engineers apply both CFD analyses and experimental, and the two complement each other. For example, engineers may obtain global properties, such as lift, drag, pressure drop, or power, experimentally, but use CFD to obtain details about the flow field, such as shear stresses, velocity and pressure profiles, cooler inlet temperature, higher fluid flow rate, material change, mixing cold plate materials, more aggressive fin configurations, rerouting fluids, or requesting a target temperature change and flow streamlines. CFD is employed to shorten the design cycle through carefully controlled parametric studies, thereby reducing the required amount of experimental testing.

CFD provides numerical approximation to the equations that govern fluid motion. Application of the CFD to analyze a fluid problem requires the following steps. First, the mathematical equations describing the fluid flow are written. These are usually a set of partial differential equations. These equations are then discretized to produce a numerical analogue of the equations. The domain is then divided into small grids or elements. Finally, the initial conditions and the boundary conditions of the specific problem are used to solve these equations. The solution method can be direct or iterative. In addition, certain control parameters are used to control the convergence, stability, and accuracy of the method.

CFD modeling plays an important role in the thermal design of cold plates. When the thermal designer has a good handle regarding the issues just listed, it is recommended that a first solution pass be made with a CFD program. The target temperature requested is compared with the predicted hottest spot, along with allowable pressure drop and the cooling fluid inlet temperature at a given flow. If any of the required parameters are not met, solutions can be sought by altering many parameters. These include possible cooler inlet temperature, higher fluid flow rate, and material change, mixing cold plate materials, more aggressive fin configurations, rerouting fluids, or requesting a target temperature change.

CFD for Air cooling

CFD analysis of an electronics cooling enclosure used in larger telecommunication radar system. An original cooling enclosure was simulated using Flotherm software which gives result as the benchmark thermal performance. It was show the operating temperature of the Radio Frequency (RF) components will exceed the design temperature limit of the PCB. A solution is redesign of thermal spreading arrangements using a 3 mm thick copper shelf and a Vapour Chamber (VC) heat pipe was found to bring the operating temperatures of all RF components within the specified temperature limits. The use of a VC, in particular, reduced the 60 W RF component steady state temperatures by an average of 5.4C. The study also shows that increasing the finned heat exchanger cooling airflow rate can lower further the RF components temperature though at the expense of increasing energy consumption of the fan. [8]

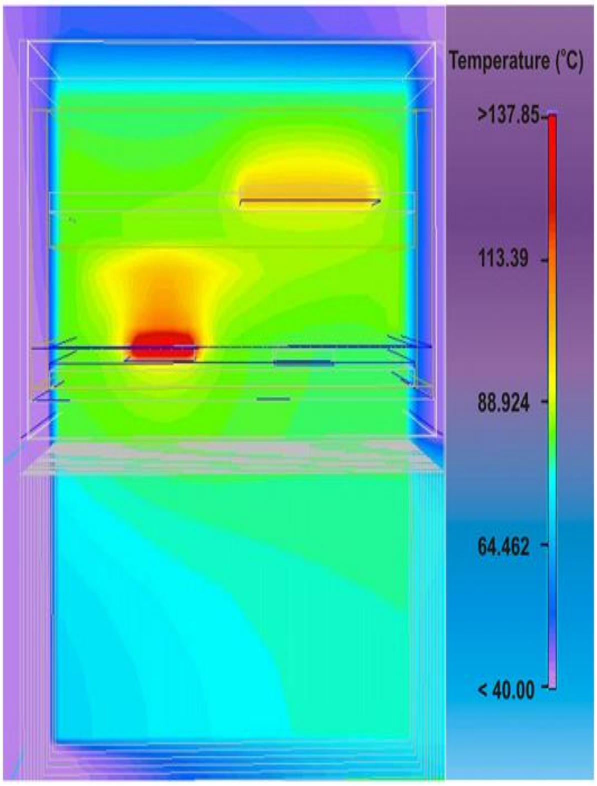


Fig. 3.Cooling enclosure side view heat [8]

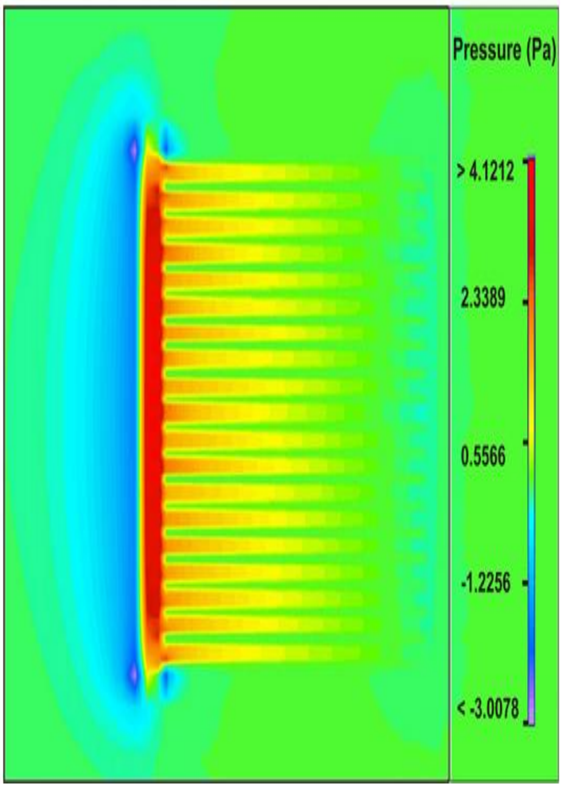


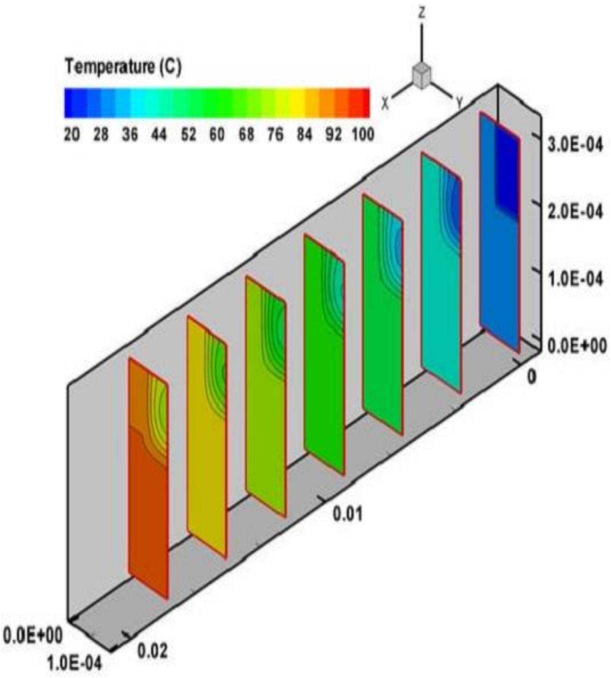
Fig. 4 Pressure distribution in the sink temperature gradient [8]

CFD for liquid cooling

The study of Flow Network Modeling (FNM) in analyzing liquid cooling systems that incorporate microchannel heat sinks. This study is done by two methods. In the first part, an analytical model of a microchannel heat sink is study and its validity is established by comparison with a detailed CFD analysis. In the second part, a liquid cooled system that accommodates three microchannel heat sinks is analyzed using Flow Network Modeling (FNM). In the FNM method, by using analytical or empirical correlations the pressure drop and heat transfer coefficient of the components in the system are calculated. The goal of the analysis is to balance the liquid flow passing through each heat sink such that the chips case temperature remains below recommended values. This study illustrates the use of the FNM technique for analyzing liquid cooled systems and microchannel heat sinks.

The CFD solution consist the calculation of the pressure and velocity fields in the rectangular microchannels. The calculated values of fRe_D agree with the analytical values of 57 and 62 (for the two aspect ratio) to four significant figures.

The CFD calculation gives the temperature distribution. Figure 5 and 6 show the temperature distribution inside the microchannel and silicon substrate for the HSAR1 and the water mass flow rate of 5 milligram/s. The temperature distribution for other cases is similar to the one shown below. [9]



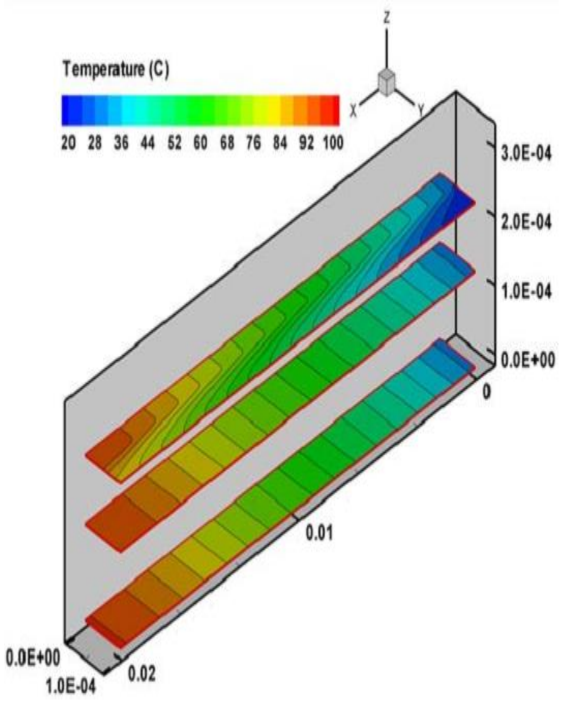


Fig.5 Temperature distribution inside HSAR1 at xy planes [9] **Fig.6 Temperature distribution inside HSAR1 at yz planes [9]**

III. TWO PHASE COOLING METHOD

Currently Global Warming Potential (GWP) & Ozone Depletion Potential (ODP) are necessary to reduce for balance environment condition. That need of achieving optimum cooling at chip level at the expense of low input pumping power. Currently, “air-cooling” and “single-phase water on-chip cooling” using copper micro-channel coolers are used but “two-phase on chip cooling” with refrigerants have a very large scale & long lasting advantages. However, due to the complexity of modelling two-phase flow, this solution is not yet well understood. Modelling of two phase flow, particularly liquid vapor under diabatic conditions inside a horizontal tube using CFD analysis is difficult with the available two phase models in FLUENT due to continuously changing flow v patterns. This study is an attempt at modelling a two-phase flow for various refrigerants for the proper prediction of pressure drop and pumping power. In the present analysis, CFD analysis of two phase flow of refrigerants inside a horizontal condenser tube of inner diameter, 0.0085m and 1.2m length is carried out using homogeneous model under adiabatic conditions. The refrigerants considered here are R134a, R407C and the newly developed Du-Pont/Honeywell R1234yf. The analysis is performed at saturation temperature to evaluate the local frictional pressure drop. Using Homogeneous model, average properties are obtained for each of the refrigerants that is considered as single phase pseudo fluid. The so obtained pressure drop data is compared with experimental data and the separated flow models available in literature. [10]

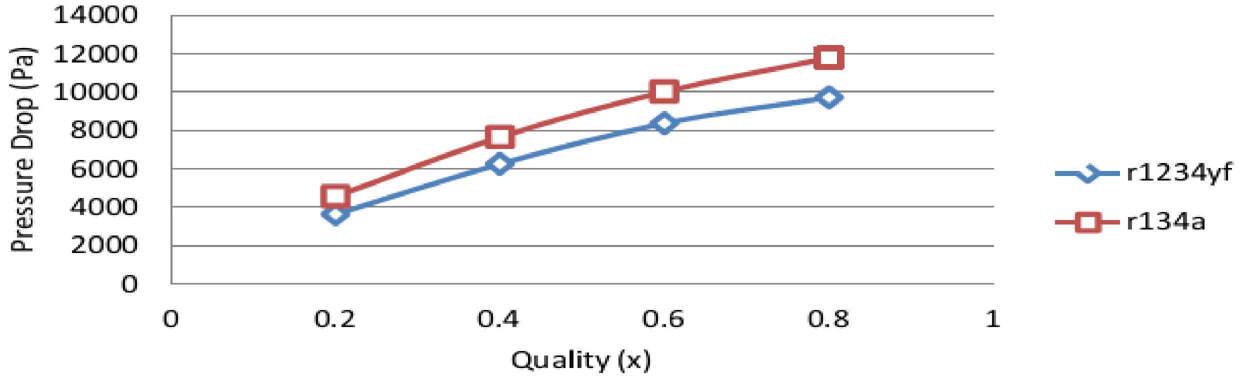


Fig.7 Pressure Drop comparison, R1234yf vs R134a, Ts = 25 degrees, G = 600kg/m^2-s [10]

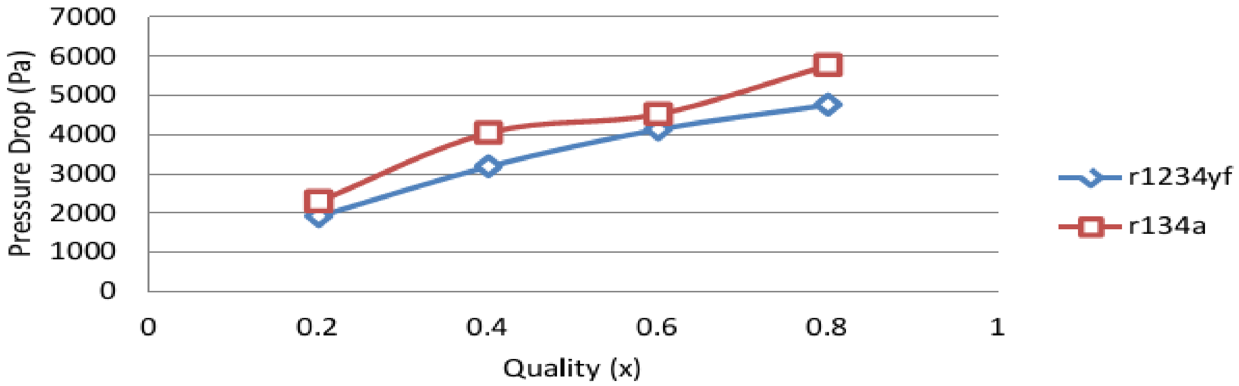


Fig.8 Pressure Drop comparison, R1234yf vs R134a, Ts = 25 degrees, G = 400kg/m^2-s [10]

IV. CONCLUSION

Based on the CFD analysis, it can be concluded that liquid cooling is a highly effective solution for managing the high heat dissipation generated by modern electronic boards and chips. Compared with conventional air cooling, liquid cooling provides superior heat transfer performance, resulting in lower operating temperatures, improved temperature uniformity, and enhanced reliability of electronic components. The CFD simulation accurately predicts temperature distribution, fluid flow, and heat transfer characteristics, enabling the optimization of cooling channel design and operating conditions before physical implementation. Effective thermal management through liquid cooling not only prevents overheating and thermal damage but also increases the performance, efficiency, and lifespan of high-power electronic systems used in data centers, medical equipment, and other heat-intensive applications. Thus, CFD-based design of liquid cooling systems offers a cost-effective and reliable approach for developing advanced thermal management solutions for next-generation electronics.

REFERENCE

[1] **The book of fluid mechanics by Yunus A.Çengel And John M.Cimbala.**

[2] **Mangesh D. Shende, Dr.Ashish Maha lle Cooling Of Electronic Equipments with Heat Sink: A Review of Literature**

[3] **SATISH G. KANDLIKAR and CLIFFORD N. HAYNER Liquid Cooled Cold Plates for Industrial High-Power Electronic Devices—Thermal Design and Manufacturing Considerations.**

[4] **Nikhil E. Chaudhari, Nishtha Vijra, T. P. Singh Computational Fluid Dynamics Analysis of Two-Phase Thermosyphon.**

[5] **P.D. Giradkar, U.S.Wankhede High Heat Flux Micro-Electronics Cooling.**

[6] **http://www.designfax.net/cms/dfx/opens/articleviewdfx.php?nid=4&bid=165& et= featurearticle&pn=02**

[7] **https://www.classle.net/book/computational-fluid-dynamics**

[8] **R. Boukhanou, A. Haddad**

[9] **A CFD analysis of an electronics cooling enclosure for application in telecommunication systems**

[10] **Amir Radmehr and Suhas V. Patankar**

[11] **A Flow Network Analysis of A Liquid Cooling System That Incorporates Microchannel Heat Sinks**

[12] **Two Phase Flow Cfd Analysis Of Refrigerants In A Condenser Pipe For Prediction Of Pressure Drop And Pumping Power By Aniket Ramchandra Kalambe**