Investigation of Heat Dissipation In Heat Pipe Using CFD

Syed Azheruddin[1], V.K.Pravin[2]
[1] Dept OF Mechanical Engineering
[2]professor, Dept OF Mechanical Engineering
[1, 2] P.D.A College Of Engineering ,Kalaburgi , Karnataka

Abstract- *Heat pipe is a simple heat transport device having very high 'effective thermal conductance'. Due to its significant advantages like compactness, light weight and passiveness, heat pipe finds a wide range of applications to dissipate heat. Some of the applications of heat pipe are cooling of electronic components, flattening of temperature in aerospace vehicles, waste heat recovery from exhaust gas, etc. Flat rectangular heat pipes can economically be employed for handling large heat transfer rates when weight and space are major constraints. The working medium which is entrapped within the heat pipe experiences phase transformations from liquid to vapour and vice versa. The vapour working medium which is formed by the vaporisation of liquid at the evaporator region by absorbing heat from the heat source, flows to the condenser section through the vapour core. The vapour condenses at the condenser zone by rejecting heat to the sink and returns to the evaporator through the porous wick structure utilising capillary pumping pressure for re-evaporation. The working fluid experiences pressure drop both in the wick region and vapour channel. The simple theory of the heat pipe states that the capillary pressure developed in the wick should be greater than the sum of the pressure drops in the vapour core and in the wick structure. If the heat pipe is not horizontal, then gravitational head will also play a role in the pressure balance.*

Keywords- Grooved heat pipe; heat pipe; thermal resistance; inclination angle; isothermal characteristics

I. INTRODUCTION

1.1 Introduction to Heat Pipe:

The field of electronics is the fast developing science and in the present scenario, its contribution to the technology is growing rapidly. Continuous usage of these devices generates high heat .This induces thermal stresses in the electronic circuits, leading to the failure in the components .The generated large heat flux is not removed effectively and it leads to deterioration in the effective functioning of the electronic devices. Also, the effective thermal management becomes one of the major serious challenges in many technologies because of constant demands for faster speed and continuous reduction of device dimensions. Heat pipe is a special type of heat exchanger that transfers large amount of heat due to the effect of capillary action and phase change heat transfer principle .It is a simple device with no moving parts that can transfer large quantities of heat over fairly large distances without requiring any power input.

1.2 Principle Of Operation:

A heat pipe is basically a sealed slender tube containing a wick structure lined on the inner surface and a small amount of fluid such as water at the saturated state. The length of the heat pipe can be divided into three parts viz. evaporator section, adiabatic section and condenser section. It is composed of three sections evaporator section, adiabatic section and condenser section

When the evaporator end of the heat pipe is brought into contact with a hot surface or placed into a hot environment, heat flows into the heat pipe. Being at a saturated state, the liquid in the evaporator end of the heat pipe vaporizes as a result of this heat transfer, causing rise in the vapour pressure. The pressure difference drives the vapour through the core of the heat pipe from the evaporator towards the condenser section. The condenser end of the heat pipe is in a cool environment, and thus, its surface is slightly cool. The vapour that comes into contact with this cooler surface of heat pipe, condensation takes place and releases a vaporized heat, which is rejected to the surrounding medium. The liquid then returns to the evaporator end of the heat pipe through the wick as a result of capillary action in the wick, in this way the cycle is completed. As a result, the heat is absorbed at one end of the heat pipe and is rejected at the other end, with thefluid inside serving as a transport medium for heat .The principle of operation of heat pipe is shown in fig 1.1

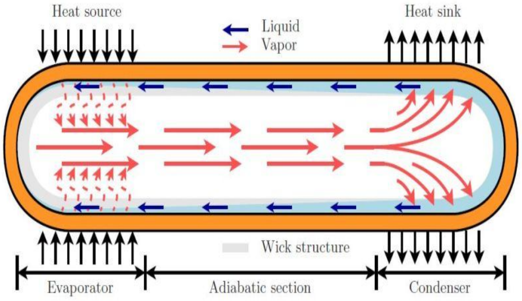


Fig 1.1 Principle of operation of heat pipe

1.3 Evolution of Heat Pipe:

The development of heat pipe originally started with Angier March Perkins who worked initially with the concept of the working fluid only in one phase (he took out a patent in 1839 on the hermetic tube boiler which works on this principle). Jacob Perkins (descendent of Angier March) patented the Perkins Tube in 1936 and they became widespread for use in locomotive boilers and baking ovens. The Perkins Tube was a system in which a long and twisted tube passed over an evaporator and a condenser, which caused the water within the tube to operate in two phases. Although these early designs for heat transfer systems relied on gravity to return the liquid to the evaporator (later called a thermo syphon), the Perkins Tube was the jumping off point for the development of the modern heat pipe. The concept of the modern heat pipe, which relied on a wicking system to transport the liquid against gravity and up to the condenser, was put forward by R.S. Gaugler of the General Motors Corporation. According to his patent in 1944, Gaugler described how his heat pipe would be applied to refrigeration systems. Heat pipe research became popular after that and many industries and labs including Los Alamos, RCA, the Joint Nuclear Research Centre in Italy, began to apply heat pipe technology in their fields. By 1969, there was a vast amount of interest on the part of NASA< Hughes, the European Space Agency, and other aircraft companies in regulating the temperature of a spacecraft and how that could be done with the help of heat pipes. There has been extensive research done regarding specific heat transfer characteristics, in addition to the analysis of various material properties and geometries.

1.4 Components Of Heat Pipe:

Heat pipe has three components:

1. Casing
2. Working Fluid
3. Wick

1.4.1 Casing:

The case is the heat pipes connection to the outside environment. Heat has to be transferred through the case to and from the working fluid in the evaporator and condenser. At the same time it is desirable to have no heat transfer in the adiabatic area and to maintain pressure differential across the walls.

Selection of the case material depends on the following factors:

- Compatibility (both working fluid and the external environment)
- Strength to weight ratio
- Thermal conductivity
- Ease of fabrication including weld ability, machinability and ductility

Materials for casing

- Aluminium
- Stainless Steel
- Copper
- Composite materials
- Refractory materials (for high temperature heat pipes) or linings to prevent corrosion

1.4.2 Working fluid:

Selection of working fluid is directly linked to the properties of the fluid. The properties are going to affect both the ability to transfer heat and comparability with the case and wick material. Following things are to be considered while choosing the working fluid

- Compatibility with wick and wall materials
- Good thermal stability
- Wet ability of wick and wall materials
- Vapour pressures not too high or low over the operating temperature range
- High latent heat
- High thermal conductivity
- Low liquid and vapour viscosities
- High surface tension

The most common working fluid is ammonia. The standard working fluid for space applications and some refrigerants is water. Meanwhile, the standard working fluid for most terrestrial applications, some organic working fluids and alkali-metals, especially sodium as the standard high temperature working fluid. The majority of heat pipes for room temperature applications uses ammonia (213-373K), methanol(283-403K), ethanol(273-403K), water(303-473K) as working fluid and typically operates in the of 20-150^{o} C.

1.4.3 Wick material/structure

The main functions of the wick are to generate capillary pressure and to distribute the liquid around the evaporator area. If the heat pipe has to return the liquid over a distance against the gravity field there is a big requirement of the wick. Therefore, there are many different forms of wick depending on the heat pipe and its location to be able to satisfy the two main functions.

Capillary Wick Designs and Structures in Heat Pipe:

The wick structure within the heat pipe is present to return condensate to the evaporator section. While small pores are needed at the liquid-vapor interface to develop high capillary pressures, large pores are preferred within the wick so that the movement of the liquid is not restricted too greatly. For this reason, many different types of wick structures have been developed in order to optimize the performance of the capillary heat pipe.

The types of wick structures can be divided into two categories: homogeneous and composite wicks. Homogeneous wicks have the benefit of being relatively simple to design, manufacture and install. Composite wicks however, can significantly increase the capillary limit of the heat pipe, but have the drawback of high manufacturing costs.

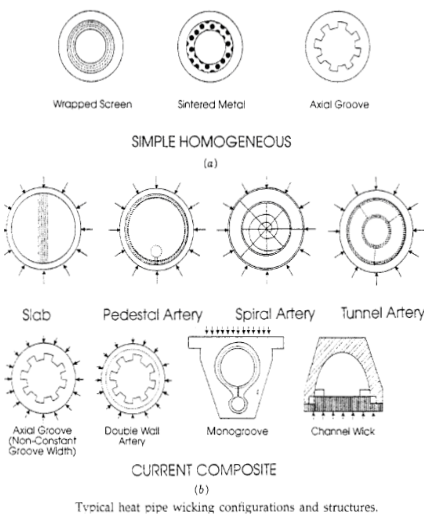


Fig 1.2 Capillary wick designs

There are three properties of wicks that are important in heat pipe design:

Minimum capillary radius:

This parameter should be small if a large capillary pressure difference is required, such as in terrestrial operation for a long heat pipe with the evaporator above the condenser, or in cases where a high heat transport capability is needed.

Permeability:

Permeability is a measure of the wick resistance to axial liquid flow. This parameter should be large in order to have a small liquid pressure drop, and therefore, higher heat transport capability.

Effective thermal conductivity:

A large value for this parameter gives a small temperature drop across the wick, which is a favorable condition in heat pipe design.

1.5 Motivation To The Present Work:

As electronic devices get smaller, engineers and designers are faced with the growing challenge of keeping up with the need to optimize processing speed within a shrinking form factor. Faster processors necessitate increased power consumption, which generates heat; and smaller form factors necessitate greater miniaturization of the implements used to disperse that heat. These considerations are forcing clever engineers and designers to think in terms of systemic solutions in which every consideration in a device’s power equation is examined for greatest optimization. In general, requirements of the power equation demand that heat dissipation must be proportional to the power dissipation of a given device. Power dissipation is the amount of electricity wasted by a device (i.e., power dissipation is dependent on the capacitance of the logic elements, the operating voltage swing, and the operating frequency). Even though the processors in cell phones, for example, often use just a few hundred milliwatts of electricity, much of this is simply lost to heat.

Conventional methods for cooling of electronic equipment include improving the design of printed circuit board, Using of thermal interface materials to fill the microscopic air gaps and using of fans .These methods are replaced by integration of heat pipes which provides efficient cooling

1.6 Scope and Objectives:

Heat pipes can be used extensively in electronics and electrical equipment's ,energy systems ,aerospace and avionics ,heat exchangers and heat pumps ,gas turbine engines and automotive industries ,production tools ,medicine and human body temperature control ,ovens and furnaces ,manufacturing ,transportation systems and de-icing .Of course there are different types of heat pipes depends on wick structure like sintered wick ,thermo syphon ,and grooved wick.

From the literature review it is understood that most of the research is going on sintered wick heat pipe only so i have selected helical grooved wick heat pipe and it is also have

i. Less weight
ii. High performance in gravity assisted condition
iii. Radial temperature drop is minimum

Very few papers are available related to prediction of thermal performance using empirical relationships incorporating process parameters. It is very useful in cooling applications. Moreover, there is no literature available on establishing relationship between thermal performance and process parameters of heat pipe .Development of such relationships will also useful to select optimum parameters which optimize the thermal performance of heat pipe .Keeping all the points in mind ,the present investigation was carried out to attain following objectives

i. Studying the effect of heat pipe process parameters(mass flow rate ,inclination angle and heat input) on the thermal performance
ii. Developing empirical relationships to predict the thermal performance of heat pipe incorporating process parameters using Design of Experiments(DOE),Analysis of Variance(ANOVA) and Regression analysis
iii. Optimizing the heat pipe process parameters to attain maximum thermal performance using Response Surface Methodology(RSM)

1.7 Types Of Heat Pipes:

Heat pipes have been designed and built with various cross - sectional areas as small as 30 µ m width × 80 µm depth and 19.75 mm in length (micro heat pipes), and heat pipes as large as 100m in length. For simplicity of design and manufacturing, heat pipe containers are generally circular cylinders. Other shapes, however, such as rectangular (flat heat pipes), conical (rotating heat pipes), corrugated flexible heat pipes, and nose cap (leading edge heat pipes) geometries have been studied.

1.7.1Two –Phase Closed Thermo syphon:

A two -phase closed thermo syphon is a gravity - assisted wickless heat pipe. The condenser section is located above the evaporator so that the condensate is returned by gravity. The sonic and vapour pressure limits are constraints to the operation of the thermo syphon as with capillary driven heat pipes. The entrainment limit is more profound in the thermo syphon than in capillary driven heat pipes due to the free liquid surface

II. INTRODUCTION OF CFD

Computational Fluid Dynamics (CFD) has grown from a mathematical curiosity to become an essential tool in almost every branch of fluid dynamics, from aerospace propulsion to weather prediction. CFD is commonly accepted as referring to the broad topic encompassing the numerical solution, by computational methods. These governing equations, which describe fluid flow, are the set of Navier-Stokes equation, continuity equation and any additional source terms, for example, porous medium or electric body force.

Since the advent of the digital computer, CFD, as a developing science, has received extensive attention throughout the international community. The attraction of the subject is twofold. Firstly, there is the desire to be able to model physical fluid phenomena that cannot be easily simulated or measured with a physical experiment, for example, weather systems. Secondly, there is desire to be able to investigate physical fluid systems more cost effectively and more rapidly than with experimental procedures.

Traditional restrictions in flow analysis and design limit the accuracy in solving and visualisation of the fluid-flow problems. This applies to both single and multi-phase flows, and is particularly true of problems that are three dimensional in nature and involve turbulence, additional source terms, and/or heat and mass transfer. All these can be considered together in the application of CFD, a powerful technique that can help to overcome many restrictions inherent in traditional analysis.

CFD is a method for solving complex fluid flow and heat transfer problems on a computer. CFD allows the study of problems that are too difficult to solve using classical techniques. The flow inside the ESP is complex and this can be analysed using CFD tool, which provides an insight into the complex flow behaviour.

III. CFD SIMULATIONS

The process of performing CFD simulations is split into three components:

- Pre-processing
- Solving.
- Post Processing
- Pre-processing

The pre-processor contains all the fluid flow inputs for a flow problem. It can be seen as a user-friendly interface and a conversion of all the input into the solver in CFD program. At this stage, quite a lot of activities are carried out before the problem is being solved. These stages are listed below:

Geometry Definition - The region of interests, that is the computational domain which has to be defined **Grid generation**- It is the process of dividing the domain into a number of smaller and non-overlapping sub-domains

.

Physical and chemical properties - The flow behaviour in terms of physical and chemical characteristics are to be selected.

Fluid property Definition - The fluid properties like density and viscosity are to be defined.

Boundary conditions - All the necessary boundary conditions have to be specified on the cell zones.

The solution of the flow problem such as temperature, velocity, pressure etc. is defined at the nodes insides each cell. The accuracy of the CFD solution is governed by the number of cells in the grid and is dependent on the fineness of the grid

Solution

In the numerical solution technique, there are three different streams that form the basis of the solver. They are finite difference, finite element and finite volume methods. The differences between them are the way in which the flow variables are approximated and the discretisation processes are done.

Finite Difference Method (FDM)

FDM describes the unknown flow variables of the flow problem by means of point samples at node points of a grid coordinate. By FDM, the Taylor’s expansion is usually used to generate finite differences approximation.

Finite Element Method (FEM)

FEM uses the simple piecewise functions valid on elements to describe the local variations of unknown flow variables. Governing equation is precisely satisfied by the exact solution of flow variables. In FEM, residuals are used to measure the errors.

Finite Volume Method (FVM)

FVM was originally developed as a special finite difference formulation. The commercial CFD code packages using the FVM approaches are PHOENICS, FLUENT, FLOW 3D and STAR-CD. Basically, the numerical algorithm in these CFD commercial packages involves the formal integration of the governing equation over all the finite control volume, the discretisation process involves the substitution of a variety of FDM types to approximate the integration equation of the flow problem, and the solution is obtained by iterative method. Discretisation in the solver involves the approaches to solve the numerical integration of the flow problem. Usually, two different approaches are made, one at a time.

Explicit approach: Usually, this is the most useful approach that makes sense. It is relatively simple to set up and program. The limitation is that for a given t and x, the time must be less than some limit imposed by stability constraints. In some cases, t must be very small to maintain the stability, and consequently long running time is required for the calculation over a given time interval t.

Implicit approach: For this approach, the stability can be maintained over a large value of t and fewer time steps are required for making calculation resulting in less computer time. But it is complicated to set up and program. The computer time per time step is much larger than the explicit approach due to the matrix manipulation, which is required for each time step. This approach is very accurate to follow the exact transients, i.e., the time variations of the independent variables.

Post-Processing

The CFD package provides the data visualisation tools to visualise the results of the flow problem. This includes – vectors plots, domain geometry and grid display, line and shaded counter plots, particle tracking etc. Recent facilities are aided with animation for dynamic result display and they also have data export facilities for further manipulation external to the code.Determining the convergence, whether the solution is consistent and stable for all range of flow variables, is important.Convergence is a property of a numerical method to produce a solution that approaches the exact solution by which the grid spacing and control volume size are reduced to a specific value or to zero value.Consistency is to produce the system of algebraic equations that can be equivalent to the original governing equation.Stability associates with the damping of errors as a numerical method proceeds. If a technique chosen is not stable, even the round-off error in the initial data can lead to wild oscillations or divergence.

IV. RESULTS AND DISCUSSION

4.1 Results for 102.9W:

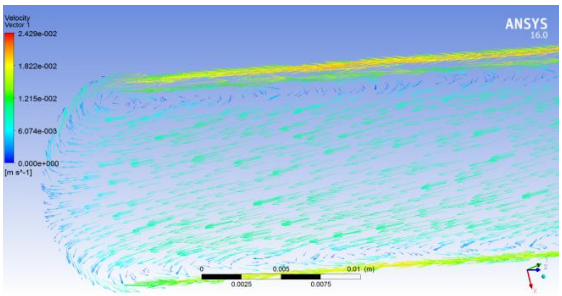


Fig 4.1. Conversion of phase and flow pattern inside the wicked tube.

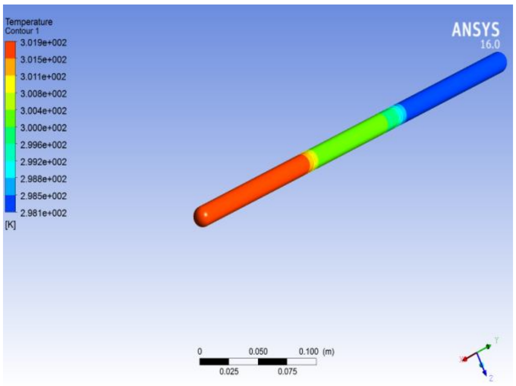


Fig 4.4 Temperature Contour.

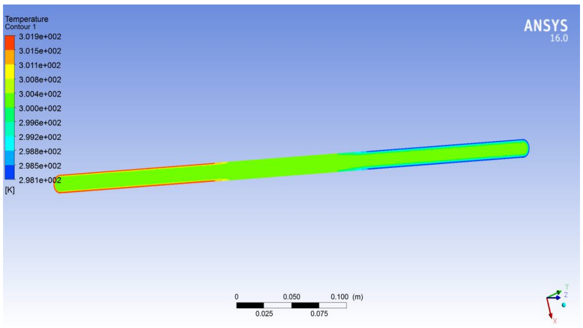


Fig 4.5 Temperature inside the tube.

4.2 Results for 150 W:

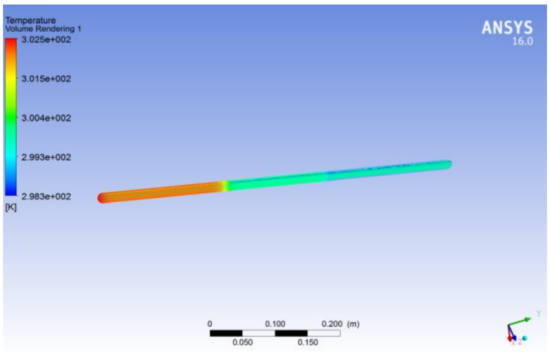


Fig 4.7 Temperature Contour 150W.

Table 4.1Inclination angle (θ) = 15^o and mass flow rate (m) = 0.01 kg/s.

4.3 Results 30 Degrees 102.9W:

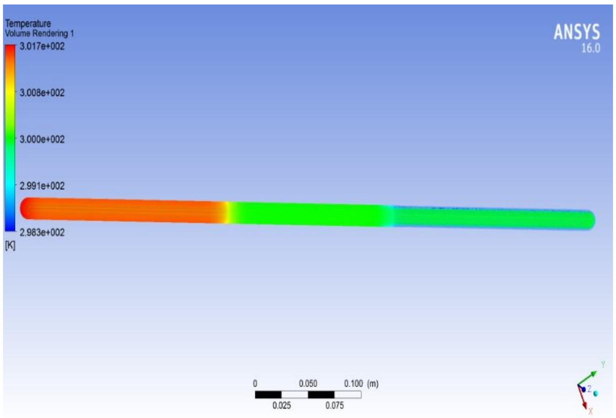


Fig 4.9 Temperature along the heat pipe.

4.4 Results 30 Degrees 150W:

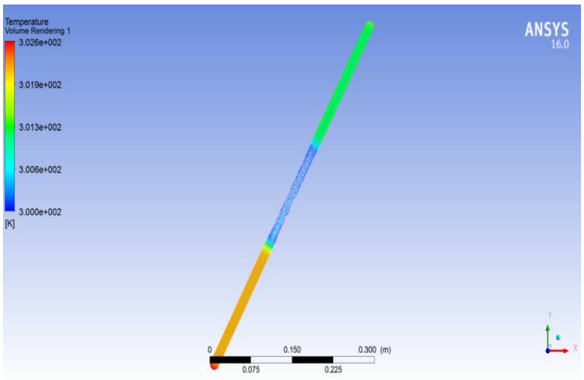


Fig 4.12 30 Degree 150W temperature Distribution.

Table 4.2Inclination angle (θ) = 30 ° and mass flow rate(m) = 0.01 kg/s

4.5 Results for 60 degrees 102.9W:

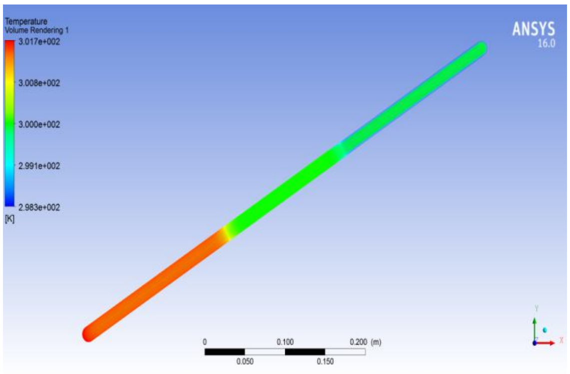


Fig 4.15 Temperature distribution in 60 degree inclination heat pipe.

4.6 RESULTS 60 Degree 150W:

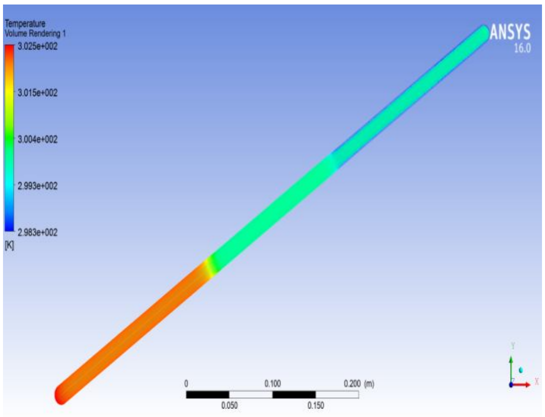


Fig 4.19 Temperature Distribution 60 degree 150W.

Table 4.3 Inclination angle (θ) = 60 ° and mass flow rate(m) = 0.01 kg/s.

4.7Results 102.9 W 90 Degrees Inclination:

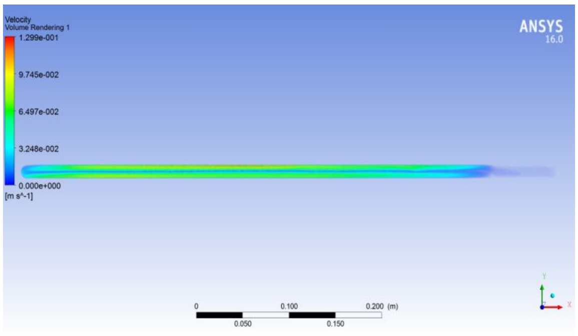


Fig 4.20 Velocity distribution 102.9W 90 Degrees.

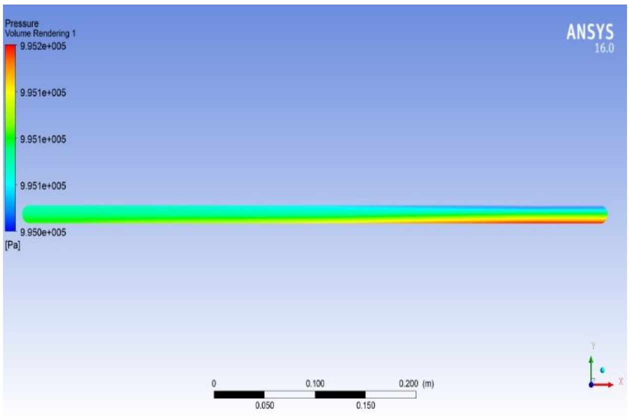


Fig 4.21 Pressure distribution of 90 Degrees 102.9W.

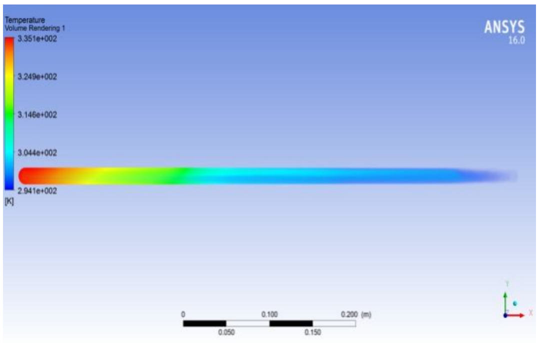


Fig 4.22 Temperature Distribution of 90 degree 102.9W.

4.8 Result 150W 90 inclination:

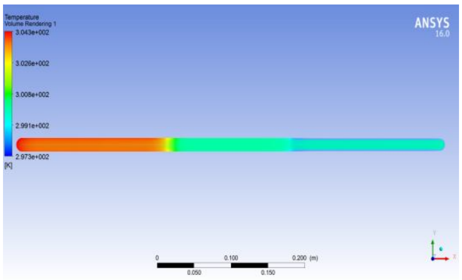


Fig 4.24 Temperature distribution in 150W 90 degrees inclination

Table 4.4 Inclination angle (θ) = 90 ° and mass flow rate (m) = 0.01 kg/s

4.9. Sample Calculations:

At Inclination Angle = 15^{o} and mass flow rate (m)= 0.01 kg/s

1) For heat input 102.9W

Rth = $\frac{Te - Tc}{Qin}$

= $\frac{(38-34)}{102.9}$

= 0.0389 K/W

$$h=\frac{Qin}{As*(Ts-Tc)}$$

$$=\frac{102.9}{0.00997*(4)}$$

$= 2585.42\ W/m^2K$

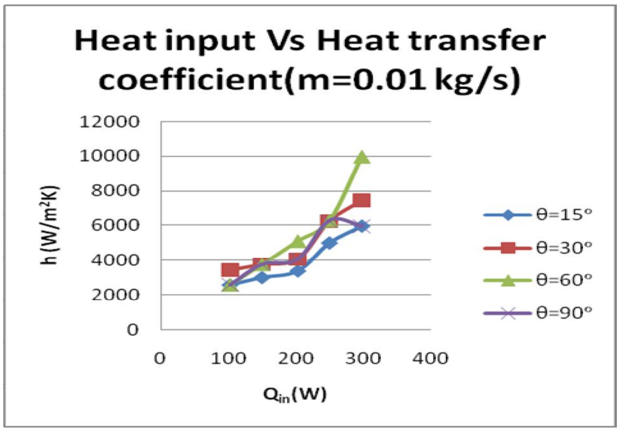
Where

$A_s = \pi * D * L_e$

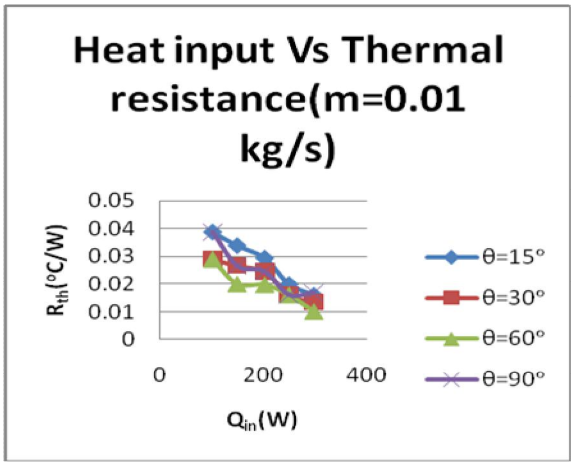
$= \pi * 0.01588*0.2$

$= 0.00997m^2$

4.10. Graphs & Explanations:



Graph 4.1 Heat input Vs Heat transfer coefficient at m=0.01 kg/s



Graph 4.2 Heat input Vs Thermal resistance at m=0.01 kg/s

From the graphs (fig 4.1 and fig 4.2) we can summarize as follows. Thermal resistance decreases with increase in heat input for any particular angle. As heat input increases continuously, more and more liquid will be vaporized so that heat can be removed more effectively by phase change. As the degree of vaporization is proportional to the amount of heat input, the thermal resistance decreases gradually until the minimum occur. Beyond the minimum point, larger input heating power will require more condensate for vaporization due to faster vaporization and faster vapour flow. The faster vapour flow create more interfacial resistance between the flow of vapour and condensate and will result in an inadequate return of the condensate to the evaporation section. Thus, the thermal resistance will increase more quickly as compared to that of the lower input heating powers. Hence the thermal resistance decreases up to 200W and then increases i.e. there will be more heat transfer enhancement at 200W heat input and after that it decreases

Thermal resistance generally decreases with increase in inclination angle as gravitational effect is more pronounced but at higher inclination angles there is no time to exchange the heat by the heat pipe to the coolant (water) at condenser section. So thermal resistance decreases up to 60° and after that it increases

Thermal resistance decreases with increase in mass flow rate of water in the condenser section
Heat transfer coefficient increases with increase in heat input up to 200W as the thermal resistance decreases and above 200W heat transfer coefficient decreases as thermal resistance increases

Heat transfer coefficient increases with increase in inclination angle up to 60° as the thermal resistance decreases and above 60° heat transfer coefficient decreases as thermal resistance increases

Heat transfer coefficient increases with increase in mass flow rate of water in the condenser section

V. CONCLUSION

- In this investigation effect of heat pipe process parameters i.e. mass flow rate (m) of coolant ,inclination angle (θ) of heat pipe and heat input power(Q) to the helical grooved heat pipe were studied.
- With increase in mass flow rate (m) of coolant, thermal resistance (R_{th}) decreases and heat transfer coefficient increases gradually, which means that thermal performance increases.
- With increase in inclination angle (θ) of heat pipe, thermal resistance decreases up to 60° and further increase of inclination angle the thermal resistance increases. With increase in inclination angle, heat transfer coefficient increases up to 60° and further increase of inclination angle the heat transfer coefficient decreases.
- With increase in heat input power (Q) to the heat pipe, thermal resistance decreases up to 150w and further increase of heat input the thermal resistance increases. With increase in heat input, heat transfer coefficient

increases up to 150 w and further increase of heat input the heat transfer coefficient decreases.

REFERENCES

[1] R.Senthilkumar ,J.Prabhakaran and M.Karthikeyan (2014) Analysis of Energy Conservation Potential of the Heat pipe using Response Surface Methodology,*International Journal of Development and Research*.ISSN:2230-9926,Vol.4,pp:570-574

[2] R.Senthilkumar,J.Prabhakaran and M.Karthikeyan(2014) Analysis Of Heat pipe Operating parameters by Response Surface Methodology using Copper Nanofluid as a Working fluid,*Global Digital Central*,ISSN:2155-658X

[3] G.Kumaresan,S.Venkatachalapathy,LazarusGodsonAsirvatham(2014) Experimental Investigation on Enhancement in Thermal Characteristics of Sintered Wick Heat pipe using CuoNanofluids, *International Journal of Heat and Mass Transfer*,Vol.72, Pp: 507-516.

[4] R.Senthilkumar,S.Vaidyanathan and B.Sivaraman(2010) Thermal Analysis of heat pipe using Taguchi method,*International Journal of Engineering Science and Technology*,Vol.2(4),pp:564-569

[5] Jung-Shun Chen and Jung-Hua Chou(2015) The Length and Bending angle Effects on the Cooling Performance of Flat plate heat pipes,*International Journal of Heat and Mass Transfer* 90(2015)848-856

[6] R.Maniraman,K.Palaniradja,N.Alagumurthi,J.Hussain(2012) Factors affecting the Thermal performance of Heat Pipe,*Journal of Engineering Research and Studies*,E-ISSN0976-7916

[7] Nandy Putra , WayanNataSeptiadi, Haolia Rahman, RidhoIrwansyah(2012) Thermal performance of Screen mesh wick Heat pipes with Nanofluids, *International Journal of Heat and Mass Transfer* ,Vol 40

[8] A.A Walnuj,F.ZPathan,A.AShaik,Ahmedkhadim Hussein(2015) Heat transfer Enhancement in Heat pipe using Nanofluid,*Ird India*,ISSN:2319-3182,Vol-4

[9] A.KMozumder,A.FAkon,M.S.HChowdhary,S.CBanik(2010) Performance of Heat pipe for different Working fluids and Fill ratios,*Journal of Mechanical Engineering*,Vol.ME 41,No.2

[10] Yaxiong Wang and G. P. Peterson(2005) Investigation of a Novel Flat heat pipe,*Journal of Heat transfer*,127(2),165-170

[11] Randeep Singh, AliakbarAkbarzadeh, Masataka Mochizuki(2009) Effect of Wick Characteristics on the Thermal Performance of the Miniature Loop Heat Pipe, *Journal of Heat transfer*,131(8),082601

[12] XueZhihu,Qu Wei(2014) Experimental study on effect of Inclination angles to Ammonia pulsating heat pipe,*Chinese Journal of Aeronautics*,27(5):1122-1127

[13] S.MPeyghambarzadeh,S.Shahpouri,N.Aslanzadeh,M.Rahimnejad(2013) Thermal performance of different working fluids in dual diameter circular heat pipe,*Ain Shams Engineering Journal*(2013)4,855-861

[14] Nishthavijra and Tejinderpaulsingh(2015) An Experimental study of Thermal performance of concentric annular heat pipe,*American International Journal of Research in Science,Technology,Engineering& Mathematics*,2328-3491