CFD STUDY ON HEAT TRANSFER CHARACTERISTICS OF A HEAT PIPE

A project report submitted in partial fulfilment of the requirement for the award of

BACHELOR OF TECHNOLOGY
IN
MECHANICAL ENGINEERING

Submitted by

HARISH BEHARA	317126520135
S. PRANEETH	317126520172
D. SAIKUMAR	317126520128
B. ROHIT	317126520124
P. SHYAM	316126520161

Under the esteemed guidance of

K. GOWRI SANKAR (Asst. Professor)

M. Tech.

Department of Mechanical Engineering

PRAGNANAM BRAHMA
ANITS

ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY & SCIENCES (A)

(Permanently affiliated to Andhra University, Approved by AICTE, Accredited by NBA & NAAC with ‘A' grade) Sangivalasa – 531162, Bheemunipatnam (Mandal), Visakhapatnam (District), Andhra Pradesh, India

ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY & SCIENCES (A)

(Affiliated to Andhra University, Approved by AICTE, Accredited by NBA & NAAC with A grade)

SANGIVALASA, VISAKHAPATNAM (District) – 531162

PRAGNANAM BRAHMA

ANITS

CERTIFICATE

This is to certify that the Project Report entitled **"CFD STUDY ON HEAT TRANSFER CHARACTERISTICS OF A HEAT PIPE"** being submitted by HARISH BEHARA (317126520135), SRIPATHI PRANEETH KUMAR (317126520172), DANGETI SUNDAR SAI KUMAR (317126520128), BELAMANA ROHITH (317126520124), PODUGU SHYAM (317126520161) in partial fulfillments for the award of degree of **BACHELOR OF TECHNOLOGY** in **MECHANICAL ENGINEERING, ANITS**. It is the work of bona-fide, carried out under the guidance and supervision of **MR.K.GOWRI SHANKAR**, Assistant Professor, Department Of Mechanical Engineering, ANITS during the academic year of 2017-2021.

PROJECT GUIDE	Approved By HEAD OF THE DEPARTMENT
(MR.K.GOWRI SHANKAR) **Assistant Professor** **Mechanical Engineering Department** **ANITS, Visakhapatnam.**	**(Dr. B. Naga Raju)** **Head of the Department** **Mechanical Engineering Department** **ANITS, Visakhapatnam.**

Department of ... Engineering
ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY & SCIENCE
Sangivalasa-531 162 VISAKHAPATNAM Dist A P

ACKNOWLEDGEMENT

We express immensely our deep sense of gratitude to **Mr. K. Gowri Sankar,** Assistant professor, Department of Mechanical Engineering, Anil Neerukonda Institute of Technology and sciences, Sangivalasa, Bheemunipatnam Mandal, Visakhapatnam district for her valuable Guidance and encouragement at every stage of the work made it a successful fulfilment.

We are very thankful to **Prof. T. V. HANUMANTHA RAO**, principal and **Prof.B.NAGARAJU**, Head of the Department, Mechanical Engineering, Anil Neerukonda Institute of Technology and sciences for providing this opportunity.

We express our sincere thanks to the members of Non-Teaching staff of Mechanical Engineering department for their kind co-operation and support to carry on work.

Last but not the least, we would like to convey our thanks to all those who directly or indirectly contributed for the completion of work.

ABSTRACT

Current work intends on development of a three-dimensional cylindrical heat pipe and simulating the performance by varying the material and working fluid based on incompressible fluid assumption. The three-dimensional asymmetric model is developed in ANSYS Design module. ANSYS fluent software is used to apply numerical procedure in coupled system of standard levels. Numerical method is known to perform very well over a wide range of fluids, heat inputs, materials. Also, the numerical method is used to validate the results of previously published numerical Heat Pipe results. A comprehensive study is carried out to illustrate effect of each parameter on thermal performance of cylindrical heat pipe with different materials and fluids. The model is used to describe heat transfer in liquid-wick zone and source by means of evaporation and condensation. This presents the heat loss due to evaporation and condensation at liquid vapor interface. The model gives the axial outer wall temperature profile, center line velocity magnitude, center line pressure of cylindrical heat pipe due to circumferential heat.

CONTENTS

LIST OF TABLES

LIST OF FIGURES

NOMENCLATURE

A = Cross section area (m^2)

k_{eff} = Effective Thermal conductivity (W/(m·K))

N = Mesh number (1/m)

Q = Heat (W)

R_{th} = Thermal Resistance(^0C/W)

t = Time (s)

T_w = Thickness of the wick (m)

$\vec{V}$ = Velocity Vector (m/s)

Greek symbols

λ = Coefficient of bulk viscosity(poise)

μ = Dynamic viscosity(kg/ms)

ρ_l = Density of liquid(kg/m^3)

ρ_v = Density of vapour(kg/m^3)

τ = Shear force(N)

ε = Wick porosity

λ = Latent heat of Vaporization (J/kg)

ν = Poisson's ratio

h = Heat transfer coefficient(W/m^2K)

σ = Surface tension(N/m)

ABBREVIATIONS

CFD - Computational fluid dynamics

VOF - Volume of Fluid

Chapter-1

INTRODUCTION

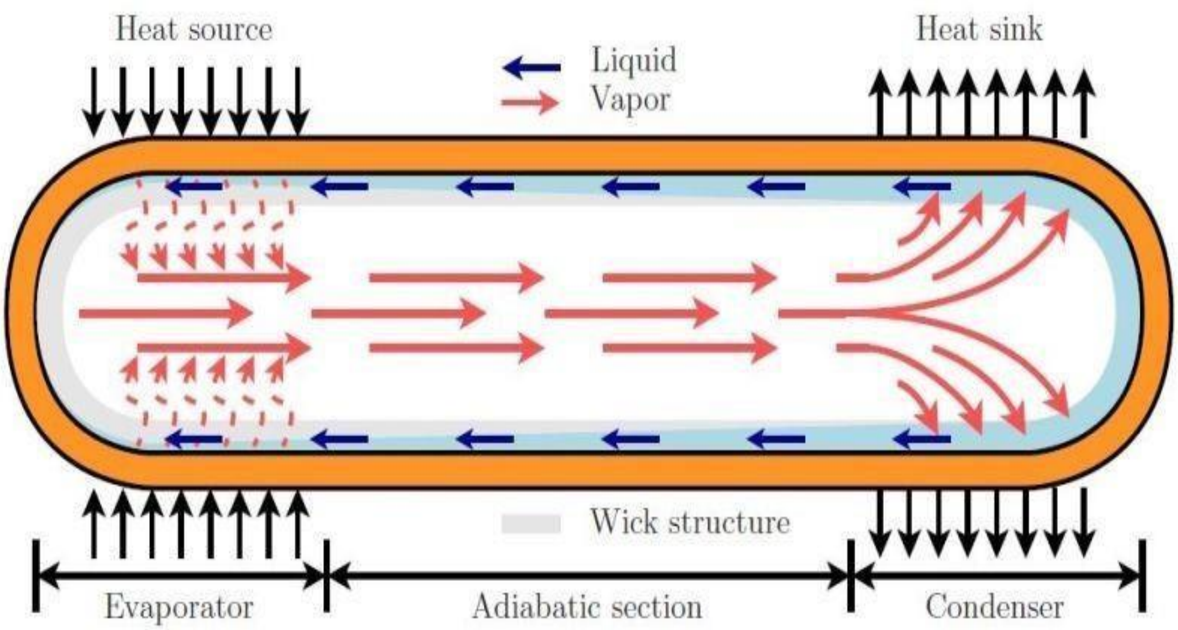


FIG 1.1 BASIC WORKING OF A HEAT PIPE

A heat pipe is a two-phase heat transfer device with a very high effective thermal conductivity. It is a vacuum tight device consisting of an envelope, a working fluid, and a wick structure. The heat input vaporizes the liquid working fluid inside the wick in the evaporator section. The saturated vapor, carrying the latent heat of vaporization, flows towards the colder condenser section. In the condenser, the vapor condenses and gives up its latent heat. The condensed liquid returns to the evaporator through the wick structure by capillary action. The phase change processes and two- phase flow circulation continues as long as the temperature gradient between the evaporator and condenserare maintained.

1.1 ROLE OF CFD IN ANALYSIS

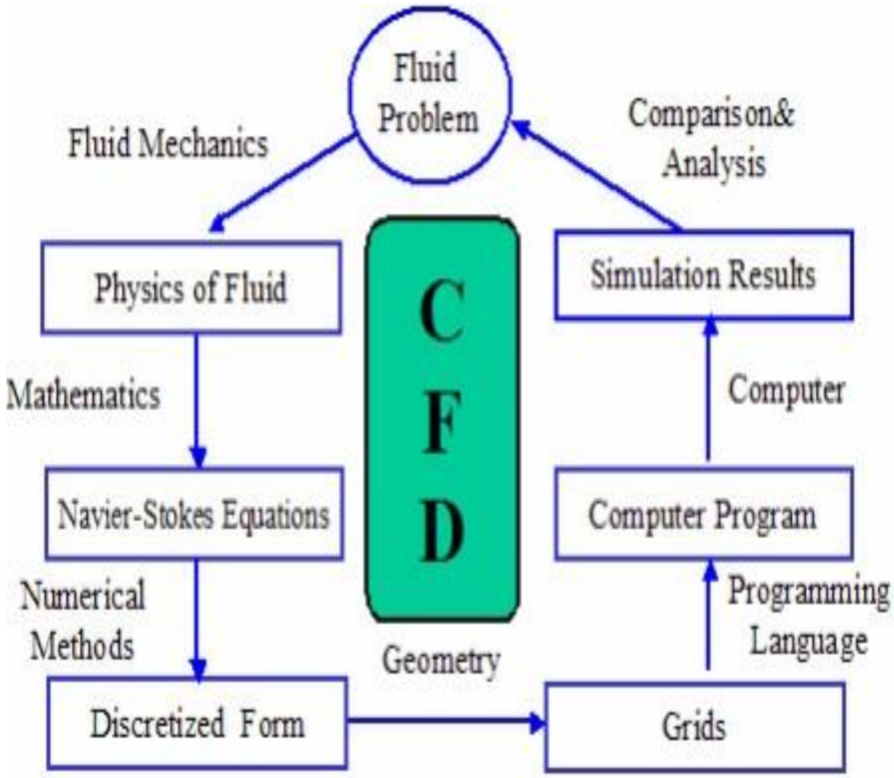


Fig1.2 Process of Computational Fluid Dynamics

CFD provides a qualitative analysis of the flow in fluids. Sometimes, however, it can also perform quantitative analysis of the fluid flow. To do this, it uses algorithms and numerical methods to solve problems regarding the fluid flow. The interaction of solids with their surrounding fluid, both liquid and gas can be simulated with the help of CFD. The boundary conditions can be determined with the help of it. With progress in research, even complex scenarios like turbulent flow can be simulated and various parameters of the fluid flow can be determined with the help of more accurate software. Navier - Stoke´s equation is the basis for most of the CFD modelling problems. These equations describe the flow of a single-phase fluid. Research led to development of more complex equations that take into account more and more of reallife parameters that affect the fluid flow. CFD utilizes partial differential equations (PDE) for mathematical modelling, discretization and solution in numerical methods and several software tools processing before and after analysis. The computer simulations are performed in a virtual flow laboratory ‘by trained scientists and engineers. The triumph of simulation over experimentation is that it is much faster and cheaper. Parallel and multi - purpose simulations can be conducted simultaneously. The simulations are most accurate when the flow to be evaluated is a laminar one instead of a turbulent flow. Similarly, it is much more difficult to simulate multi - phase flows than to simulate single phase flows. The accuracy improves manifold if the fluid is chemically inert instead of being a reactive flow. Working Principle of CFD - Firstly, a mathematical model of the physical problem is constructed with the help of equations. Care is taken so that the equations satisfy the conservation of momentum, energy and mass.

Some of the properties are determined empirically. Even then some assumptions have to be made to simplify the problem. Usually, it is considered that the fluid is flowing with a steady flow rate. Secondly, numerical methods are applied to these equations.

This is called discretization and it employs methods like Finite element method, finite volume method, finite difference method etc. These equations are solved simultaneously for very small volumes of the fluid known as mesh. This involves iterations and re - iterations using a computer until and unless a convergence is reached. Thirdly, after processing, parameters like drag friction, torque, heat transfer, pressure loss can be determined. In the real life, CFD has been used to design aerodynamic shapes, the shape and flow.

1.2 BASIC COMPONENTS OF A HEAT PIPE

1. The container
2. The working fluid
3. The wick or capillary structure

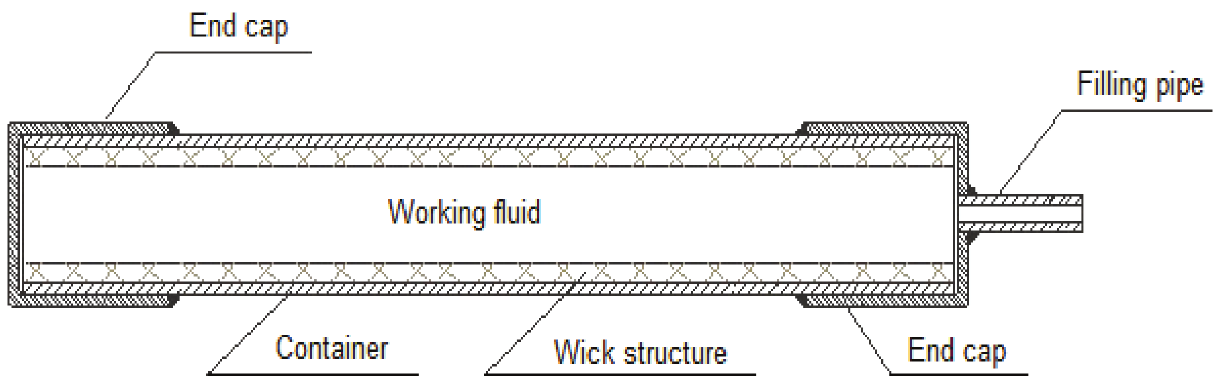


Fig1.3 Basic components of Heat Pipe

1.2.1 CONTAINER

The function of the container is to isolate the working fluid from the outside environment. Selection of the container material depends on many factors. These are as follows:

1. Compatibility with working fluid and external environment
2. Strength to weight ratio
3. Thermal conductivity
4. Porosity

1.2.2 WORKING FLUID

The first consideration in selecting the working fluid is the operating vapor temperature range. Within the approximate temperature range, several working fluids may exist and many characteristics should be examined in order to determine the most suitable working fluid for the application.

1.2.3 WICK OR CAPILLARY STRUCTURE

The wick structure in a heat pipe facilitates liquid return to the evaporator from the condenser. The main purposes of wick are to generate the capillary pressure, and to distribute the liquid around the evaporator section of heat pipe. The commonly used wick structure is a wrapped screen wick.

1. The wick is a porous structure made of materials like steel, aluminium, nickel or copper in various ranges of pore sizes

2. The prime purpose of the wick is to generate capillary pressure to transport the working fluid from the condenser to the evaporator.

3. It must also be able to distribute the liquid around the evaporator section to any area where heat is likely to be received by the heat pipe.

4. Wicks are fabricated using metal foams, and more particularly felts, the latter being more frequently used. By varying the pressure on the felt during assembly, various pore sizes can be produced.

5. The maximum capillary head generated by a wick increases with decrease in pore size.

6. The wick permeability increases with increasing pore size.

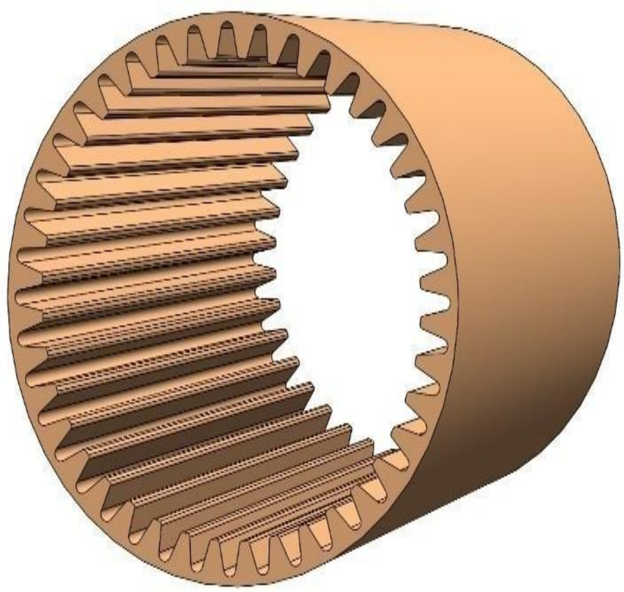


Fig1.4 Grooved wick structure

1.3 CONSTRUCTION OF HEAT PIPE

A general heat pipe consists of a sealed hollow tube, which is made from a thermo-conductive metal such as copper or aluminum. The pipe contains a small quantity of "working fluid" (such as water, ethanol or mercury) with the remaining of the pipe being filled with vapor phase of the working fluid. On the internal side of the tube's side-walls a wick structure exerts a capillary force on the liquid phase of the working fluid. This is typically a sintered metal powder (sintering is a method for making objects from powder, by heating the material until its particles adhere to each other) or a series of grooves etched in the tube's inner surface. The basic idea of the wick is to soak up the coolant.

Heat pipes contain no moving parts and require no maintenance and are completely noiseless. In theory, it is possible that gasses may diffuse through the pipe's walls over time, thus reducing this effectiveness. The vast majority of heat pipes use either ammonia or water as working fluid. Extreme applications may call for different materials, such as liquid helium (for low temperature applications) or mercury (for extreme high temperature applications). The advantage of heat pipe is their great efficiency in transferring heat. They are actually a better heat conductor than a mass of solid copper.

As mentioned previously there is liquid vapor equilibrium inside the heat pipe. When thermal energy is supplied to the evaporator, this equilibrium breaks down as the working fluid evaporates. The generated vapor is at a higher pressure than the section through the vapor space provided. Vapor condenses giving away its latent heat of vaporization to the heat sink. The capillary pressure created in the menisci of the wick, pumps the condensed fluid back to the evaporator section. The cycle repeats and the thermal energy is continuously transported from the evaporator to condenser in the form of latent heat of vaporization. When the thermal energy is applied to the evaporator, the liquid recedes into the pores of the wick and thus the menisci at the liquid-vapor interface are highly curved.

1.4 TYPES OF HEAT PIPE

In addition to Standard and Constant Conductance Heat Pipes (CCHPs), there are a number of other types of heat pipes. Including:

1. Vapor Chambers (flat heat pipes), which are used for heat flux transformation, and iso-thermalization of surfaces.

2. Variable Conductance Heat Pipes (VCHPs), which use a Non-Condensable Gas (NCG) to change the heat pipe effective thermal conductivity as power or the heat sink conditions change

3. Pressure Controlled Heat Pipes (PCHPs), which are a VCHP where the volume of the reservoir, or the NCG mass can be changed, to give more precise temperature control

4. Diode Heat Pipes, which have a high thermal conductivity in the forward direction, and a low thermal conductivity in the reverse direction

5. Thermosyphons, which are heat pipes where the liquid is returned to the evaporator by gravitational/acceleration forces,

6. Rotating heat pipes, where the liquid is returned to the evaporator by centrifugal forces.

1.4.1 VAPOR CHAMBER HEAT PIPES OR FLAT HEAT PIPES

Thin planar heat pipes have the same primary components as tubular heat pipes: a hermetically sealed hollow vessel, a working fluid, and a closed-loop capillary recirculation system. In addition, a series of posts are generally used in a vapor chamber, to prevent collapse of the flat top and bottom when the pressure is lower than atmospheric, which is 100 °C for water vapor chambers.

There are two main applications for vapor chambers. First, they are used when high powers and heat fluxes are applied to a relatively small evaporator. Heat input to the evaporator vaporizes liquid, which flows in two dimensions to the condenser surfaces. After the vapor condenses on the condenser surfaces, capillary forces in the wick return the condensate to the evaporator. Note that most vapor chambers are insensitive to gravity, and will still operate when inverted, with the evaporator above the condenser. In this application, the vapor chamber acts as a heat flux transformer, cooling a high heat flux from an electronic chip or laser diode, and transforming it to a lower heat flux that can be removed by natural or forced convection. With special evaporator wicks, vapor chambers can remove 2000 W over 4 cm2 or 700 W over 1 cm2.

Second, compared to a one-dimensional tubular heat pipe, the width of a two- dimensional heat pipe allows an adequate cross section for heat flow even with a very thin device. These thin planar heat pipes are finding their way into "height sensitive" applications, such as notebook computers and surface mount circuit board cores.

1.4.2 VARIABLE CONDUCTANCE HEAT PIPES (VCHPS)

Standard heat pipes are constant conductance devices, where the heat pipe operating temperature is set by the source and sink temperatures, the thermal resistances from the source to the heat pipe, and the thermal resistances from the heat pipe to the sink. In these heat pipes, the temperature drops linearly as the power or condenser temperature is reduced. For some applications, such as satellite or research balloon thermal control, the electronics will be overcooled at low powers, or at the low sink temperatures. Variable Conductance Heat Pipes (VCHPs) are used to passively maintain the temperature of the electronics being cooled as power and sink conditions change. VCHPs have two additions compared to a standard heat pipe: 1. A reservoir and 2. A Non-Condensable Gas (NCG) added to the heat pipe, in addition to the working fluid; see the picture in the Spacecraft section below. This NCG is typically argon for standard VCHPs and helium for Thermosyphons. When the heat pipe is not operating, the NCG and working fluid vapor are mixed throughout the heat pipe vapor space. When the VCHP is operating, the NCG is swept toward the condenser end of the heat pipe by the flow of the working fluid vapor. Most of the NCG is located in the reservoir, while the remainder blocks a portion of the heat pipe condenser. The VCHP works by varying the active length of the condenser. When the power or heat sink temperature is increased, the heat pipe vapor temperature and pressure increase.

The increased vapor pressure forces more of the NCG into the reservoir, increasing the active condenser length and the heat pipe conductance. Conversely, when the power or heat sink temperature is decreased, the heat pipe vapor temperature and pressure decrease, and the NCG expand, reducing the active condenser length and heat pipe conductance.

The addition of a small heater on the reservoir, with the power controlled by the evaporator temperature, will allow thermal control of roughly ±1-2 °C. In one example, the evaporator temperature was maintained in a ±1.65 °C control band, as power was varied from 72 to 150 W, and heat sink temperature varied from +15 °C to -65 °C. Pressure Controlled Heat Pipes (PCHPs) can be used when tighter temperature control is required. In a PCHP, the evaporator temperature is used to either vary the reservoir volume, or the amount of NCG in the heat pipe. PCHPs have shown mille-

Kelvin temperature control.

1.4.3 LOOP HEAT PIPE

A loop heat pipe (LHP) is a passive two-phase transfer device related to the heat pipe. It can carry higher power over longer distances by having co-current liquid and vapor flow, in contrast to the counter-current flow in a heat pipe. This allows the wick in a loop heat pipe to be required only in the evaporator and compensation chamber. Micro loop heat pipes have been developed and successfully employed in a wide sphere of applications both on the ground and in space.

1.5Applications

The heat pipe has been, and currently is being, studied for a wide variety of applications, covering almost the complete spectrum of temperatures encountered in heat transfer processes. The applications range from the use of liquid helium heat pipes to aid target cooling in particle accelerators, to cooling systems for state-of-the-art nuclear reactors and potential developments aimed at new measuring techniques for the temperature range 2000 ºC 3000 ºC.

Broad Areas of Application

In general, the applications come within a number of broad groups, each of which describes a property of the heat pipe. These groups are:

- Separation of heat source and sink
- Temperature flattening
- Heat flux transformation
- Temperature control
- Thermal diodes and switches

The high effective thermal conductivity of a heat pipe enables heat to be transferred at high efficiency over considerable distances. In many applications where component cooling is required, it may be inconvenient or undesirable thermally to dissipate the heat via a heat sink or radiator located immediately adjacent to the component. For example, heat dissipation from a high power device within a module containing other temperature sensitive components would be effected by using the heat pipe to connect the component to a remote heat sink located outside the module. Thermal insulation could minimize heat losses from intermediate sections of the heat pipe.

The second property listed above, temperature flattening, is closely related to source sink separation. As a heat pipe, by its nature, tends towards operation at a uniform temperature, it may be used to reduce thermal gradients between unevenly heated areas of body. The body may be the outer skin of a satellite, part of which is facing the sun, the cooler section being in shadow. Alternatively, an array of electronic components mounted on a single pipe would tend to be subjected to feedback from the heat pipe, creating temperature equalization.

The third property listed above, heat flux transformation, has attractions in reactor technology. In thermionic, for example, the transformation of a comparatively low heat flux, as generated by radioactive isotopes, into sufficiently high heat fluxes capable of being utilized effectively in thermionic generators has been attempted

The fourth area of application, temperature control, is best carried out using the variable conductance heat pipe. This can be used to control accurately the temperature of devices mounted on the heat pipe evaporator section.

While the variable conductance heat pipe found its first major applications in many more mundane applications, ranging from temperature control in electronics equipment to ovens and furnaces.

1.6 ADVANTAGES

- Passive heat exchange with no moving parts.
- Relatively space efficient.
- The cooling or heating equipment size can be reduced in some cases.
- The moisture removal capacity of existing cooling equipment can be improved.
- No cross-contamination between air streams.

1.7 DISADVANTAGES

- High cost.
- Requires the two air streams to be adjacent to each other.
- Requires that air streams must be relatively clean and may require filtration.

Chapter- 2

LITERATURE SURVEY

Aloke Kumar Mozumder et.al. [1] in their study, Heat pipe experiment was conducted both with working fluid and without fluid. Three different working fluids are taken such as distilled water, methanol, and acetone which have varying useful working range of temperature are tested with also different fill ratios. Results shows that the operating temperature of the evaporator of a heat pipe should be such that the evaporation of the working fluid can occur in it and in the condenser; the condensation occurs in it. This is important, because, without phase change, heat pipe cannot be used as an effective device for enhancement of heat transfer. At 85% fill ratio, heat transfer is the maximum remaining other experimental conditions constant. For this reason, 85% fill ratio can be regarded as an optimum fill ratio. This study reveals that the dominating parameters for the heat transfer of a heat pipe are saturated boiling temperature of the liquid, evaporator surface temperature, latent heat of vaporization of the working fluids, and fill ratio.

K Snehith and S. Bhanu praksah [2] in their study, an attempt is made to design, fabricate and test a copper heat pipe. Experiments were conducted with and without working fluid for different inclinations to assess the thermal performance of heat pipe. The working fluids chosen for the study are acetone and distilled water. The thermal performance of the heat pipe was quantified in terms of thermal resistance and overall heat transfer coefficient by measuring temperature distribution across the heat pipe. Results shows that the copper heat pipe is found to be effective when acetone is used as working fluid. The optimum inclination angle of heat pipe for maximum rate of heat transfer is found to be 60^0 for both the working fluids tested and concluded as heat pipe selected showed a better thermal performance with wet run when compared to dry run.

Leonard L. Vasiliev [3] in this study, heat pipes are used as modern heat exchangers to evaluate its performance. Heat pipes are passive, highly reliable and offer high heat transfer rates and concluded as a short review of heat pipe R&D contains mainly data from FSU which testifies, that heat pipes are very efficient heat transfer devices, which can be easily implemented as thermal links and heat exchangers in different systems to ensure the energy saving and environmental protection.

Xiaoming Huanga et.al. [4] this study describes the cooling problems in electronic equipment withhigh heat flux but small area under natural convection, a heat pipe sink with a novel fin array of unequal and continuously changing heights is proposed to improve heat transfer by reducing flowresistance. The results show that the fin arrays with larger height difference have lower flow resistance and higher local heat transfer coefficient. Parameter studies show that increasing the fin spacing and the maximum fin height difference can greatly reduce the material cost per unit power. the optimization technique was integrated with the Non-Dominated Sorting Genetic Algorithm II (NSGAII) and the Technique for Order Preference by Similarity to Ideal Solution (TOPSIS) in order to improve the search speed and accuracy of Pareto solutions. By analyzing the Pareto solutions obtained, it is found that the influence of fin height and fin spacingon the system performance is related rather than independent.

H.Johuara et.al. [5] In this paper they discussed about the state of the art applications, material and performance of current heat pipe devices. The paper is about four main topics, they are- low temperature heat pipes, high temperature heat pipes, thermal modelling of heat pipes and discussion. In the low and high temperature sections they discussed about suitable working fluids and operating temperatures, along with their compatibility with casing materials. Furthermore, the sections focus on some of the most widespread industrial applications, such as solar, Nano-particles, Rankine cycles, nuclear, thermoelectric modules and ceramics, in which heat pipe technologies offer many key advantages over conventional practices. The third section consists of a thorough analysis of the thermal modelling side of heat pipes. Internal and external thermal modelling techniques, theories and methodologies are discussed in this section, for various applications such as non-Newtonian fluids, Nano-fluids, solar, geothermal, automotive, hybrid storage and nuclear systems. In the final part of the paper they discussed about the limitations of heat pipes and the reasons why they are not implemented in more aspects of our lives. Operational limitations, cost concerns and the lack of detailed theoretical and simulation analysis of heat pipes are some of the point covered in this section.

Aditya Narayan et.al. [6] In this paper they discussed about development and testing of flexible heat pipes, so that the evaporator section of the heat pipe, which is connected to the heat load does not get stressed or deformed under the mounting conditions. The flexibility will prevent damage to the heat pipe and the system on which it is mounted. The flexibility is provided in the adiabatic section of the heat pipe by using a flexible metallic bellow to allow relative movement between the two ends of the heat pipe. They used a stainless steel bellow as the flexible element and brazed to pipes on both sides to form the outer casing. The flexible heat pipe has been tested in vertical heater-down position and horizontal position. As per the required specifications, the heat pipe is expected to cater to about 100 W of thermal load dissipation, in the operating range of 10°C to 100°C. They used a heat pipe of internal diameter 10 mm and 270 mm long. They found that the developed flexible heat pipe performs well in deformed configuration from an IR image of adiabatic section and also observed that the heat pipe can cope with sudden power inputs and position change.

Stephane Lipsa et.al. [7] In this paper they discussed about heat pipe studies during the period 2010-2015. The heat pipe advancements in the past 5 years are
1) understanding of evaporation and condensation phenomena on a capillary scale has been improved 2) New fluids and new materials have been successfully tested and enabled to increase the performance of heat pipes 3) Major progresses in the understanding of LHPs(Loop Heat Pipe) have now enabled to develop this technology at an industrial scale. 4) There are also good advances in Pulsating Heat Pipes (PHP). Not only these advancements but the researchers also noted some scientific questions which are to be answered. To answer those questions more studies are required.

Huei Chu Weng et.al. [8] In this paper the researchers studied about the heat transfer performance of gravity heat pipes with anodic aluminium oxide (AAO) wall surface. The main purpose is to study the effects of the length and diameter of AAO nanotubes on the temperature distribution, overall thermal resistance, and dry-out occurrence of gravity heat pipes charged with acetone under different input heat powers. The AAO nano tubes are prepared by anodizing the inner wall surface of the evaporator section of aluminium alloy gravity heat pipes. The experimental results have shown that increasing the AAO nanotube length could result in reduced temperature variation between the evaporator section and the condenser section. This has led to reduction in thermal resistance. The increase in the AAO diameter also led to the decrease in the temperature variation and overall thermal resistance. The researchers finally concluded that, if the anodic oxidation treatment is applied to the inner wall surface of the evaporator section of a gravity heat pipe, its heat transfer performance could be significantly improved. This study can be used for cooling purposes in wide range of applications.

B.Borgmeyer et.al. [9] In this paper they designed a mathematical model predicting the oscillating motion in an oscillating heat pipe. They considered a vapour bubble as the gas spring for the oscillating motions including effects of operating temperature, nonlinear vapour bulk modulus, and temperature difference between the evaporator and the condenser. Combining the oscillating motion predicted by the model, they developed a mathematical model predicting the temperature difference between the evaporator and the condenser. To verify the mathematical model, they conducted an experiment on copper oscillating heat pipe with eight turns. The test results have indicated that when the input power or the temperature difference from the evaporating section to the condensing section was higher than this onset value the oscillating motion started, resulting in an enhancement of the heat transfer in the oscillating heat pipe.

Daehoon Lee et.al. [10]A novel flexible and thin flat-plate heat pipe is fabricated and its thermal performance is characterized experimentally .A single-layered copper woven mesh with nanostructured super hydrophilic surface is used as the liquid wicking structure, whereas a triple-layered coarse mesh with bare copper surface serves for vapor transportation and mechanical support against the high vacuum pressure .It has been shown that the nanostructured super hydrophilic surface can bring about a significant enhancement in the thermal performance compared with that with conventional base copper surface. Based on the results from the experiment and the analysis, the effect of wick structure on the thermal performance of the heat pipe is discussed.

Davoud Jafari et.al. [11] This paper examines the so-called capillary performance of a freeform porous structure fabricated by advanced 3D metal printing technology. The fabricated structure is intended as wick for two-phase heat transfer devices, in which it contributes to the transport of a liquid working fluid through capillary forces. A stainless-steel porous structure is additively manufactured and characterized in terms of its porosity (ε), effective pore radius (r_{eff}), liquid permeability (K) and capillary performance (K/r_{eff}). Capillary penetration experiments are performed by means of height-time (h-t) and weight-time (w-t) techniques with different fluids to characterize the capillary performance of the printed wicks. The Kozeny–Carman correlation is found to predict the experimental values of permeability at lower flow velocities (0.07 m/s corresponding to a Reynolds number of 0.95), while at higher velocities an under-prediction of the experimental data is observed.

Shen-Chun Wu et.al.[12] The work develops a double- layer wick to improve the structural strength and transfer of the working fluid of a biporous wick under higher heat flux. The properties of double-layer wick different structures are utilized to enhance heat transfer performance in a loop heat pipe (LHP). The outer layer of the double-layer wick is the biporous wick, which utilizes large and small pores as channels through which to transport vapor and working fluid, respectively. Experimental results demonstrate that at 85 °C, the limiting temperature of the evaporator wall, the maximum heat load of the double-layer wick reached 700 W, and the heat transfer coefficient of the evaporator was 116 kW/m^2 °C. The minimum thermal resistance of the system was 0.08 °C/W. A comparison with the heat transfer performance of a monoporous wick demonstrates that the double-layer wick has a 67% higher maximum heat load (vs. 420 W for the monoporous wick) and an almost half lower total thermal resistance (vs. 0.17 °C/W for the monoporous wick).

Nampon Sangpab et.al.[13] The heat pipes are made of oxygen free copper for the container and nitrogen is used as the working fluid. The wick was composite wick made from sintered copper powder and grooved copper pipe. The original cylindrical copper heat pipe was 6 mm in outer diameter and 200 mm in length. The heat pipes with the bending radius of 21 mm and the bending angle of 0°, 30°, 60° and 90° and the final thickness of 2.5, 3.0, 4.0 mm were constructed and tested. The operating temperature of heat pipe was at approximately 78.25 ± 0.10 K. It was found that the performance of the heat pipe in terms of thermal resistances depends on both bending angle and final thickness. Both bending and flattening affect the thermal resistance by increasing it. The bending increased the thermal resistance from 0.88 to 1.07 K/W (6 mm, 0° to 6 mm, 90°) and the flattening increased the thermal resistance from 0.88 to 2.24 K/W (6 mm, 0° to 2.5 mm, 0°). And the combined effect of bending and flattening increased the thermal resistance from 0.88 to 1.56 K/W (6 mm, 0° to 3 mm, 90°)

Hai Tang et.al.[14] To increase the heat transfer capability at anti-gravity orientations, heat pipes with homogeneous and axially graded wicks were experimentally investigated. The homogeneous wick (0.8 mm thickness) sintered with 125–150 μm (coarse) copper powder operated at 100 W without dryout at horizontal orientation, while that sintered with 75–97 μm (fine) copper powder had lower critical heat load about 80 W. At a tilted angle of 90°, the critical heat load increased from 35 W to 45 W when increasing thickness from 0.8 mm to 1.1 mm and it further increased to 96 W when increasing thickness to 1.35 mm.

FeiXin et.al.[15] Mini-grooves flat heat pipe is suitable to be used for small space electronics rapid cooling due to its high thermal conductivity, simple structure and good temperature uniformity. In this paper, an axial fluid flow and heat transfer mathematical model of mini-grooves flat heat pipe in steady state is built to study the axial distributions of key parameters such as pressure, velocity and wall temperature. a novel graded-grooves wick design is put forward for flat heat pipe, which can increase the effective thermal conductivity coefficient by 12.4% with heat transfer rate 5.0 W. And the grooves wick in slope type with simple fabrication accords with practical application.

Ixue Zhang et.al.[16] A loop heat pipe (LHP) with a carbon fiber capillary wick was taken as the research object to study the steady state inside the LHP and validate the effectiveness of proposed improvements through CFD numerical simulations. A 3D model of the LHP was built and numerical simulations of steady state operation were conducted by ANSYS software. Through comparison of experiment and simulation results, heat leakage from the heated surface to the compensation chamber (CC) was identified as the major cause of failure in the feasibility experiment.The simulation shows that a silicon sheet with low thermal conductivity can increasethe liquid volume fraction in the CC to 23%, while an yttria-stabilized zirconia (YSZ) heat insulation layer can raise the figure to 74%. According to simulation results, the combination of YSZ layer and alumina-copper coating can realize a temperature drop of 5.7 K comparing with the combination of silicon sheet and copper plating.

Y.Wang et.al. [17]**,** in this work, An experimental investigation of the thermal performance of a flat plate heat pipe is presented. In order to describe the transient performance of a heat pipe, the concept of heat pipe time constant is introduced. The temperature distribution and heat transfer coefficient are obtained. An empirical correlation for the maximum temperature difference within the heat pipe as well as for the maximum temperature rise both in terms of input heatarepresented. Finally, steadystateexperimental results arecompared withtheanalytical result. it can be seen that the temperature was quite uniform on the largest outside surface of condenser wall. For the outside surface of the evaporator, where the input power is applied, the temperature variation is small. This is another favourable feature of a flat plate heat pipe as compared to a conventional heat pipe. This feature can be used to remove hot spots produced by arrays of heaters, or to design an efficient radiator. The response time to an input power is an important characteristic of a heat pipe. In this regard, the idea of a heat pipe time constant, was utilized in this work.

Jeevan J aidi et .al. [18] in their study , the thermal characteristics of a copper heat pipe using Computational Fluid Dynamics (CFD) numerical simulation are performed for transient two-phase flow and heat transfer using ANSYS fluent CFD solver. And an investigation of fill ratio at given heat input has been analyzed. An optimum fill ratio for a given heat flux is determined by comparing the thermal resistance values.

Chapter-3

MODELLING AND PREPROCESSOR SETUP OF HEAT PIPE

This project is based on finding the parameters which affect the efficiency of heat pipe, a device that transfers heat from one point to another by evaporation and condensation processes. Nowadays the main drawback of heat pipe is cost and effectiveness. The main goal of this project is to vary the different parameters like fluid, wick structure and find which of them affects the performance of heat pipe to large extent. This whole process is done in ANSYS Workbench by creating a heat pipe model with specific dimensions and material. The input values are taken from the previous journals on the heat pipe. From the journals the inputs are taken and evaluated in ANSYS Workbench. The input values are varied and the results are checked for any improvement in the performance of heat pipe.

3.1 COMPUTATIONAL FLUID DYNAMICS

Computational fluid dynamics is the science of predicting fluid flow, heat transfer, mass transfer and related phenomenon by solving the mathematical equations that govern these processes using a numerical algorithm on a computer. The technique is very powerful and spans a wide range of industrial and non-industrial applications.

3.2 ANSYS FLUENT

The advancement of computing facilities has led to the development of advanced software packages and tools for solving various practical engineering problems. One such advancement is the development of various computational fluid dynamic (CFD) software with different numerical solver methods. These computational methods are identified as suitable tools for solving various engineering problems. They also have various advantages over the traditional physical modelling. One such CFD software tool is ANSYS Fluent.

ANSYS Fluent provides comprehensive modelling capabilities for a wide range of incompressible and compressible, laminar and turbulent fluid flow problems. Steady-state or transient analyses can be performed Examples of ANSYS Fluent applications include laminar non- Newtonian flows in process equipment; conjugate heat transfer in turbomachinery and automotive engine components; pulverized coal combustion in utility boilers; external aerodynamics; flow through compressors, pumps, and fans; and multiphase flows in bubble columns and fluidized beds.

To permit modelling of fluid flow and related transport phenomena in industrial equipment and processes, various useful features are provided. These include porous media, lumped parameter (fan and heat exchanger), stream wise-periodic flow and heat transfer, swirl, and moving reference frame models. The

moving reference frame family of models includes the ability to model single or multiple reference frames. A time-accurate sliding mesh method, useful for modelling multiple stages in turbomachinery applications, for example, is also provided, along with the mixing plane model for computing time-averaged flow fields.

Another very useful group of models in ANSYS Fluent is the set of free surface and multiphase flow models. These can be used for analysis of gas-liquid, gas-solid, liquid-solid, and gas-liquid-solid flows. For these types of problems, ANSYS Fluent provides the volume-of-fluid (VOF), mixture, and Eulerian models, as well as the discrete phase model (DPM). The DPM performs Lagrangian trajectory calculations for dispersed phases (particles, droplets, or bubbles), including coupling with the continuous phase. Examples of multiphase flows include channel flows, sprays, sedimentation, separation, and cavitation. Robust and accurate turbulence models are a vital component of the ANSYS Fluent suite of models. The turbulence models provided have a broad range of applicability, and they include the effects of other physical phenomena, such as buoyancy and compressibility. Particular care has been devoted to addressing issues of near-wall accuracy via the use of extended wall functions and zonal models. Various modes of heat transfer can be modelled, including natural, forced, and mixed convection with or without conjugate heat transfer, porous media, and so on. The set of radiation models and related sub-models for modelling participating media are general and can take into account the complications of combustion.

A particular strength of ANSYS Fluent is its ability to model combustion phenomena using a variety of models, including eddy dissipation and probability density function models. A host of other models that are very useful for reacting flow applications are also available, including coal and droplet combustion, surface reaction, and pollutant formation models.

For all flows, ANSYS Fluent solves conservation equations for mass and momentum. For flows involving heat transfer or compressibility, an additional equation for energy conservation is solved. For flows involving species mixing or reactions, a species conservation equation is solved or, if the non-premixed combustion model is used, conservation equations for the mixture fraction and its variance are solved. Additional transport equations are also solved when the flow is turbulent.

3.3 GOVERNING EQUATIONS OF FLUID FLOW

The governing equations of fluid flow represent mathematical statement of the conservation laws of physics. Each individual governing equations represents a conservation principle. The fundamental equations of fluid dynamics are based on the following universal laws of conservation.

They are,

- Conservation of Mass
- Conservation of Momentum
- Conservation of Energy

3.3.1 CONTINUITY EQUATION

The equation based on the principle of conservation of mass is called continuity equation. The conservation of mass law applied to a fluid passing through infinitesimal, fixed control volume yields the following equation of continuity.

$$\frac{\partial \rho}{\partial t} + \nabla . \left(\rho \vec{V} \right) = 0$$

or equally

$$\frac{D\rho}{Dt} + \rho \left(\nabla . \vec{V} \right) = 0$$

$$\frac{DA}{Dt} = \frac{\partial A}{\partial t} + \left(\vec{V} . \nabla \right) A$$

3.3.2 MOMENTUM EQUATION

Equation for the conservation of linear momentum is also known as the Navier-Stokes equation (In CFD literature the term Navier-Stokes is usually used to include both momentum and continuity equations, and even energy equation sometimes). It is possible to write it in many different forms. One possibility is

$$\rho\frac{D\vec{V}}{Dt}=-\nabla p+\nabla.\bar{\bar{\tau}}+\rho\vec{f}$$

In order to be able to use an Eulerian description, material derivative at the left hand side, which is the acceleration vector, can be replaced with the sum of local and convective accelerations to get

$$\rho\left[\frac{\partial\vec{V}}{\partial t}+\left(\vec{V}.V\right)\vec{V}\right]=-\nabla p+\nabla.\bar{\bar{\tau}}+\rho\vec{f}$$

where is the body force per unit mass. If the weight of the fluid is the only body force, we can replace with the gravitational acceleration vector g

of the above equation is the viscous stress tensor. For Newtonian fluids viscous stresses only depend on the velocity gradient and the dependency is linear. Also it is known that needs to be symmetric in order to satisfy the conservation of angular momentum. For a Newtonian fluid the relation between and the velocity components is as follows

$$\tau_{ij}=\mu\left(\frac{\partial V_i}{\partial x_j}+\frac{\partial V_j}{\partial x_i}\right)+\lambda\left(\nabla.\vec{V}\right)\delta_{ij}$$

Where x denotes mutually perpendicular coordinate directions, μ is the dynamic viscosity and λ is known as the coefficient of bulk viscosity. It is related to the viscosity through the Stokes' hypothesis

$$\lambda+\frac{2}{3}\mu=0$$

3.3.3 ENERGY EQUATION

The energy equation is based on principle of conservation of energy. It is derived from first law of thermodynamics. The equation is as follows,

$$\rho\left[\frac{\partial h}{\partial t}+\nabla.\left(hV\right)\right]=-\frac{Dp}{Dt}+\nabla.\left(k.\nabla T\right)+\phi$$

3.4 MULTIPHASE FLOW IN FLUENT

A large number of flows encountered in nature and technology are a mixture of phases. Physical phases of matter are gas, liquid, and solid, but the concept of phase in a multiphase flow system is applied in a broader sense. In multiphase flow, a phase can be defined as an identifiable class of material that has a particular inertial response to and interaction with the flow and the potential field in which it is immersed. For example, different-sized solid particles of the same material can be treated as different phases because each collection of particles with the same size will have a similar dynamical response to the flow field.

3.4.1 Multiphase Flow Regimes

Multiphase flow regimes can be grouped into four categories: gas-liquid or liquid-liquid flows; gas-solid flows; liquid-solid flows; and three-phase flows.

3.4.1.1 GAS-LIQUID OR LIQUID-LIQUID FLOWS

The following regimes are gas-liquid or liquid-liquid flows:

- Bubbly flow: This is the flow of discrete gaseous or fluid bubbles in a continuous fluid.
- Droplet flow: This is the flow of discrete fluid droplets in a continuous gas.
- Slug flow: This is the flow of large bubbles in a continuous fluid.
- Stratified/free-surface flow: This is the flow of immiscible fluids separated by a clearly-defined interface.

3.4.1.2 GAS-SOLID FLOWS

The following regimes are gas-solid flows:

- Particle-laden flow: This is flow of discrete particles in a continuous gas.
- Pneumatic transport: This is a flow pattern that depends on factors such as solid loading, Reynolds numbers, and particle properties. Typical patterns are dune flow, slug flow, and homogeneous flow.
- Fluidized bed: This consists of a vessel containing particles, into which a gas is introduced through a distributor. The gas rising through the bed suspends the particles. Depending on the gas flow rate, bubbles appear and rise through the bed, intensifying the mixing within the bed.

3.4.1.3 LIQUID-SOLID FLOWS

The following regimes are liquid-solid flows:

- Slurry flow: This flow is the transport of particles in liquids. The fundamental behaviour of liquid-solid flows varies with the properties of the solid particles relative to those of the liquid. In slurry flows, the Stokes number is normally less than one. When the Stokes number is greater than one, the characteristic of the flow is liquid-solid fluidization.
- Hydro transport: This describes densely-distributed solid particles in a continuous liquid.
- Sedimentation: This describes a tall column initially containing a uniform dispersed mixture of particles. At the bottom, the particles will slow down and form a sludge layer. At the top, a clear interface will appear,and in the middle a constant settling zone will exist.

3.4.1.4 THREE-PHASE FLOWS

Three-phase flows are combinations of the other flow regimes listed in the previous sections.

3.4.2 EXAMPLES OF MULTIPHASE SYSTEMS

Specific examples of each regime described in are listed below:

- Bubbly flow examples include absorbers, aeration, air lift pumps, cavitation, evaporators, flotation, and scrubbers.

- Droplet flow examples include absorbers, atomizers, combustors, cryogenic pumping, dryers, evaporation, gas cooling, and scrubbers.
- Slug flow examples include large bubble motion in pipes or tanks.
- Stratified/free-surface flow examples include sloshing in offshore separator devices and boiling and condensation in nuclear reactors.
- Particle-laden flow examples include cyclone separators, air classifiers, dust collectors, and dust-laden environmental flows.
- Pneumatic transport examples include transport of cement, grains, and metal powders.
- Fluidized bed examples include fluidized bed reactors and circulating fluidized beds.
- Slurry flow examples include slurry transport and mineral processing
- Hydro transport examples include mineral processing and biomedical and physiochemical fluid systems
- Sedimentation examples include mineral processing.

3.5 VOLUME OF FLUID (VOF) MODEL THEORY

The VOF model can model two or more immiscible fluids by solving a single set of momentum equations and tracking the volume fraction of each of the fluids throughout the domain. Typical applications include the prediction of jet breakup, the motion of large bubbles in a liquid, the motion of liquid after a dam break, and the steady or transient tracking of any liquid-gas interface.

3.5.1 LIMITATIONS OF THE VOF MODEL

The following restrictions apply to the VOF model in ANSYS Fluent:

- You must use the pressure-based solver. The VOF model is not available with the density-based solver.

- All control volumes must be filled with either a single fluid phase or a combination of phases. The VOF model does not allow for void regions where no fluid of any type is present.

- Only one of the phases can be defined as a compressible ideal gas. There is no limitation on using compressible liquids using user-defined functions.
- Stream-wise periodic flow (either specified mass flow rate or specified pressure drop) cannot be modelled when the VOF model is used.

- The second-order implicit time-stepping formulation cannot be used with the VOF explicit scheme.
- When tracking particles in parallel, the DPM model cannot be used with the VOF model if the shared memory option is enabled (Parallel Processing for the Discrete Phase Model in the User's Guide). (Note that using the message passing option, when running in parallel, enables the compatibility of all multiphase flow models with the DPM model.)

3.5.2 STEADY-STATE AND TRANSIENT VOF CALCULATIONS

The VOF formulation in ANSYS Fluent is generally used to compute a time dependent solution, but for problems in which you are concerned only with a steady-state solution, it is possible to perform a steady-state calculation. A steady-state VOF calculation is sensible only when your solution is independent of the initial conditions and there are distinct inflow boundaries for the individual phases. For example, since the shape of the free surface inside a rotating cup depends on the initial level of the fluid, such a problem must be solved using the time-dependent formulation. On the other hand, the flow of water in a channel with a region of air on top and a separate air inlet can be solved with the steady-state formulation. The VOF formulation relies on the fact that two or more fluids (or phases) are not interpenetrating. For each additional phase that you add to your model, a variable is introduced: the volume fraction of the phase in the computational cell. In each control volume, the volume fractions of all phases sum to unity. The fields for all variables and properties are shared by the phases and represent volume-averaged values, as long as the volume fraction of each of the phases is known at each location. Thus, the variables and properties in any given cell are either purely representative of one of the phases, or representative of a mixture of the phases, depending upon the volume fraction values.

3.5.3 TIME DEPENDENCE

For time-dependent VOF calculations, is solved using an explicit time-marching scheme. ANSYS Fluent automatically refines the time step for the integration of the volume fraction equation, but you can influence this time step calculation by modifying the Courant number. You can choose to update the volume fraction once for each time step, or once for each iteration within each time step. These options are discussed in more detail in Setting Time-Dependent Parameters for the VOF Model in the User's Guide.

3.6 K-EPSILON TURBULENCE MODEL

K-epsilon (k-ε) turbulence model is the most common model used in Computational Fluid Dynamics (CFD) to simulate mean flow characteristics for turbulent flow conditions. It is a two equation model that gives a general description of turbulence by means of two transport equations (PDEs). The original force behind developing the K-epsilon model was to improve the mixing-length model, as well as to find an alternative to algebraically prescribing turbulent length scales in moderate to high complexity flows. K-ε model focuses on the mechanisms that affect the turbulent kinetic energy.

The exact k-ε equations contain many unknown and unmeasurable terms. For a much more practical approach, the standard k-ε turbulence model is used which is based on our best understanding of the relevant processes, thus minimizing unknowns and presenting a set of equations which can be applied to a large number of turbulent applications.

For turbulent kinetic energy k

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial(\rho k u_i)}{\partial x_i}=\frac{\partial}{\partial x_j}\left[\frac{\mu_t}{\sigma_k}\frac{\partial k}{\partial x_j}\right]+2\mu_t E_{ij}E_{ij}-\rho\varepsilon$$

This model is the most widely used and validated turbulence model with applications ranging from industrial to environmental flows, which explains its popularity. It is usually useful for free-shear layer flows with relatively small pressure gradients as well as in confined flows where the Reynolds shear stresses are most important. It can also be stated as the simplest turbulence model for which only initial and/or boundary conditions needs to be supplied.

In our study we have used the K-Epsilon method because, in heat pipe the liquid will be in two states i.e., vapor and liquid. To solve this model, two equations are required which is provided by the K-epsilon model.

Two-equation turbulence models allow the determination of both, a turbulent length and time scale by solving two separate transport equations. The standard k- Ɛ model in ANSYS Fluent falls within this class of models.

3.7. PHYSICAL CHARACTERISTICS AND ASSUMPTIONS

The heat pipe analyzed here is a simple one. The analysis is carried out using a 3D model. Three phases are utilized: liquid water, water vapor and air.

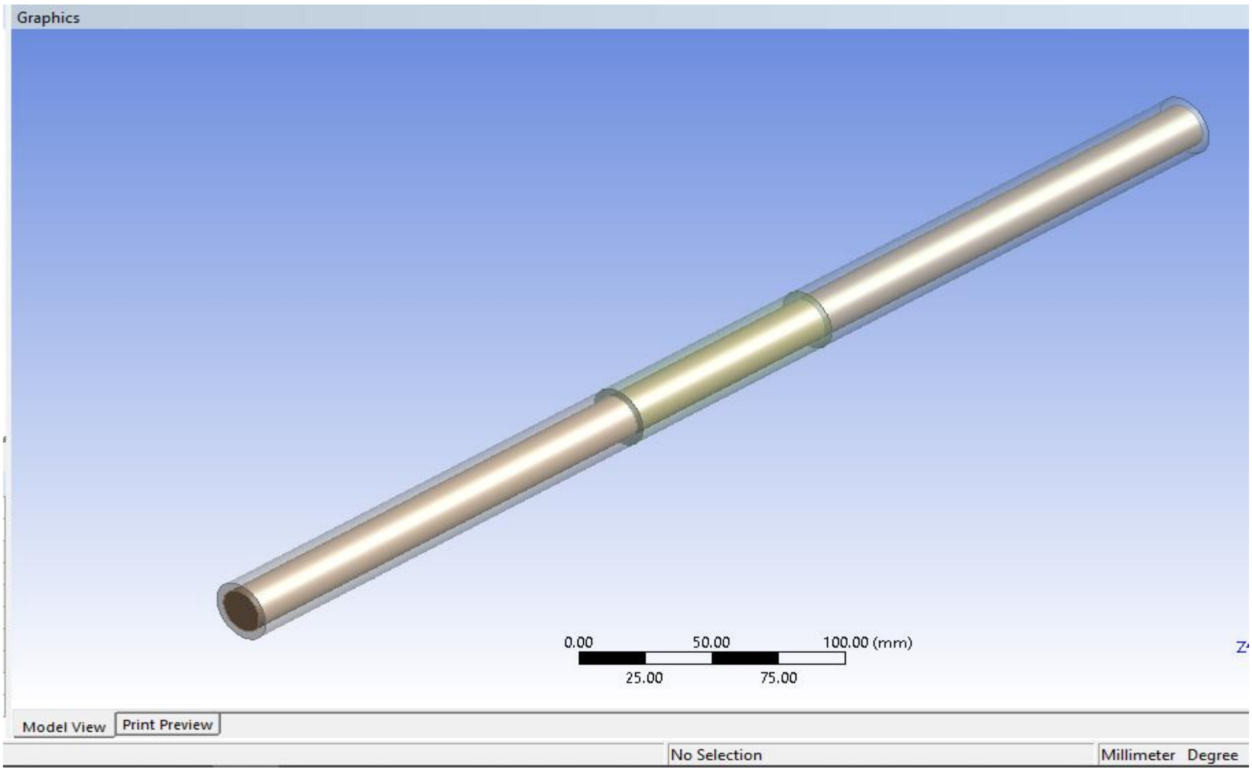


Fig 3.1 Heat pipe geometry

3.7.1 BOUNDARY CONDITIONS

The boundary conditions are defined as. The heat flux is very high because the walls are too thin. The basic process of a HP is that, when the water heats up at the bottom part of the heat pipe, it increases its temperature, vaporizing at around °C (because there is a vacuum inside the pipe), and consequently rises up. When it rises all the way up it touches the condenser and its temperature decreases, making it condense and now it goes down again.

3.7.2 ANSYS SIMULATION

The next sessions show a step by step procedure of how the simulation is done. Most of the steps come with some physical insight explaining the reason of using specific features.

3.7.2.1 GENERAL PROPERTIES

The transient model was chosen since there is a dependence on time and there is no steady state for a heat pipe. The gravity was set as 9.81 m/s² in the positive z-direction. Nothing else was changed from the default.

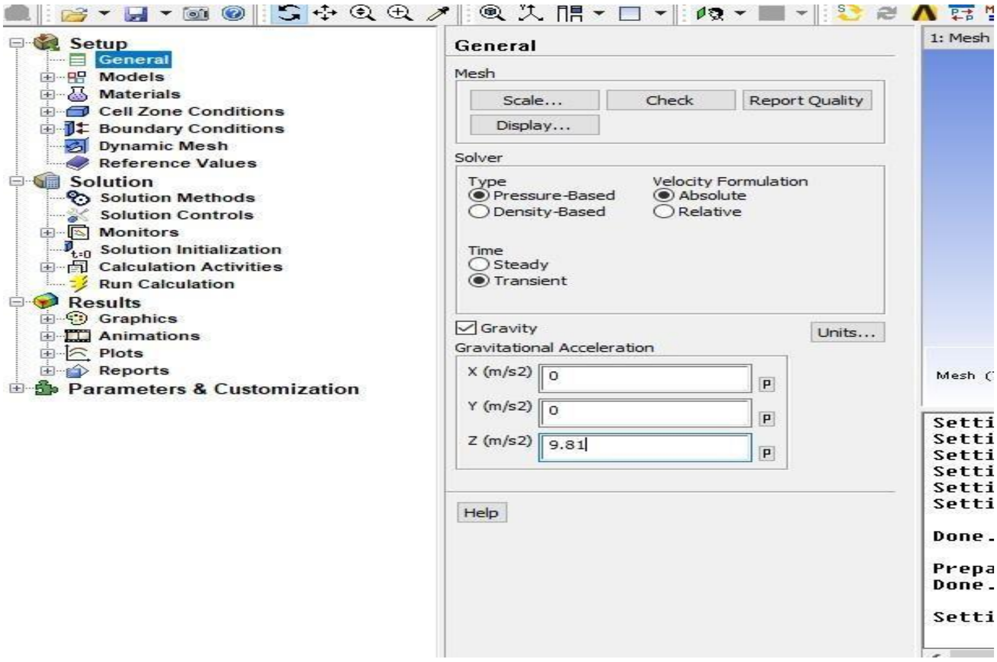


Fig 3.2 General properties of the model

3.7.2.2 SETTING UP THE MODELS

The flow inside a heat pipe is called slug flow which is characterized for a liquid–gas flow in which the gas phase exists as large bubbles separated by liquid "slugs". For this kind of flow the VOF (volume of fluid) approach is the most adequate because it tracks the interface of the phases, which is very important to describe this flow.

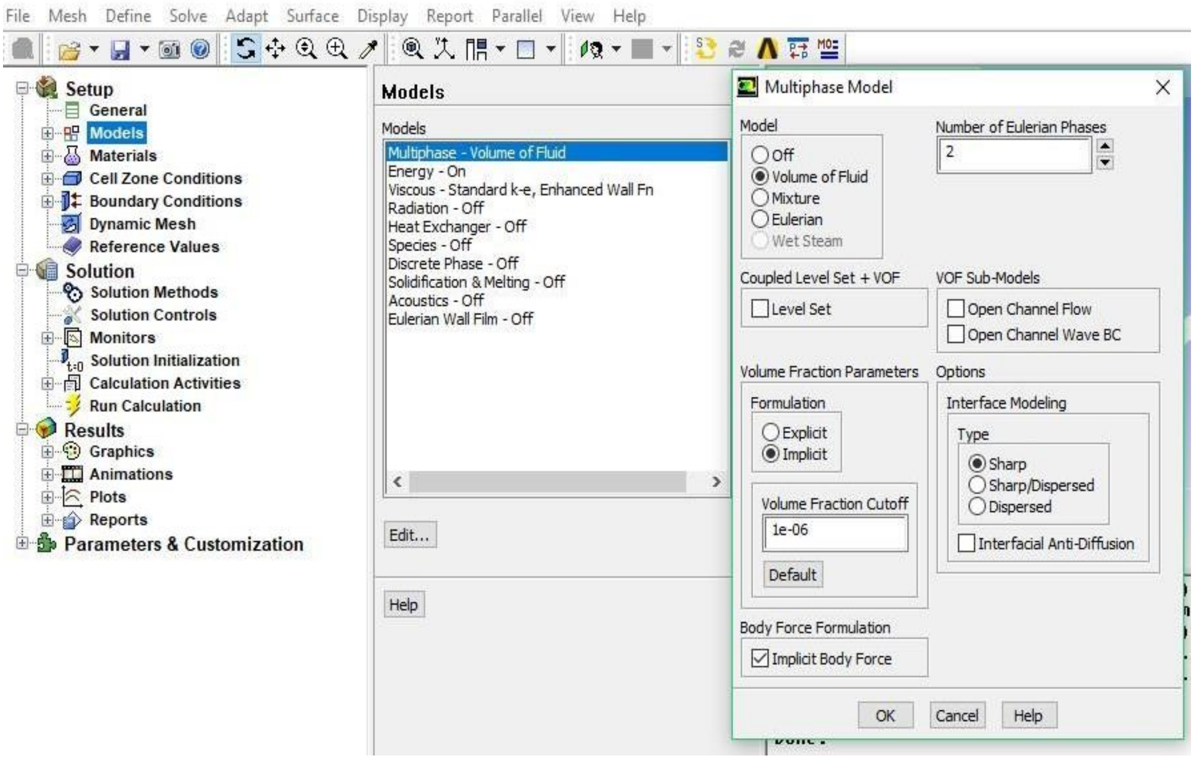


Fig 3.3 Models utilized for the analysis

Clicking on multiphase twice, the properties are set. The three phases are air, water vapor and water liquid, even though there is a vacuum inside the pipe, which is defined later, air is still defined as one of the phases. The reason for including a phase with air is that if only water liquid and vapor are defined, the calculation starts as if there is already water vapor inside the pipe. The implicit Body Force box was checked as well.

The energy equation was turned on to allow heat transfer between the phases and the flow is laminar with the viscous heating box checked. The viscous heating is turned because it includes the effect of transformation of friction on the walls into heat. To show that the flow is laminar a conservative assumption is made. It assumes a velocity of 0.1 m/s.

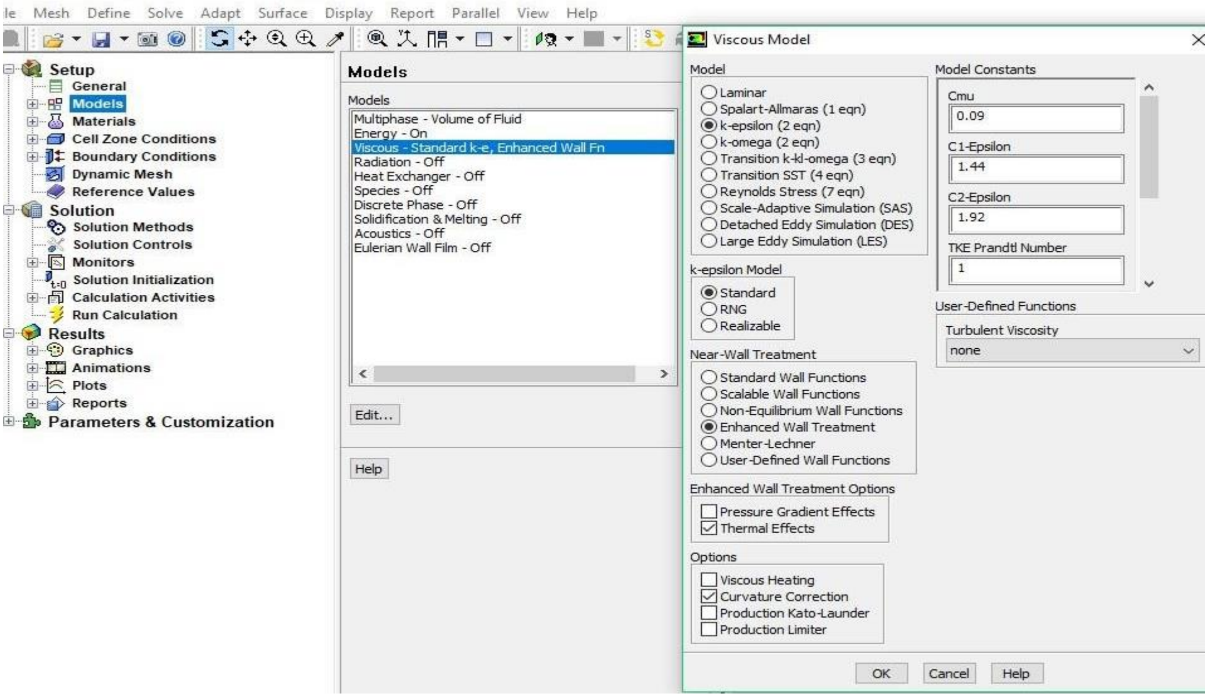


Fig 3.4 Defining the viscous model

3.7.2.3 IMPORTING THE MATERIALS

The next step is to import the materials needed from the fluent library. To do that, click on materials and then on Create/Edit.

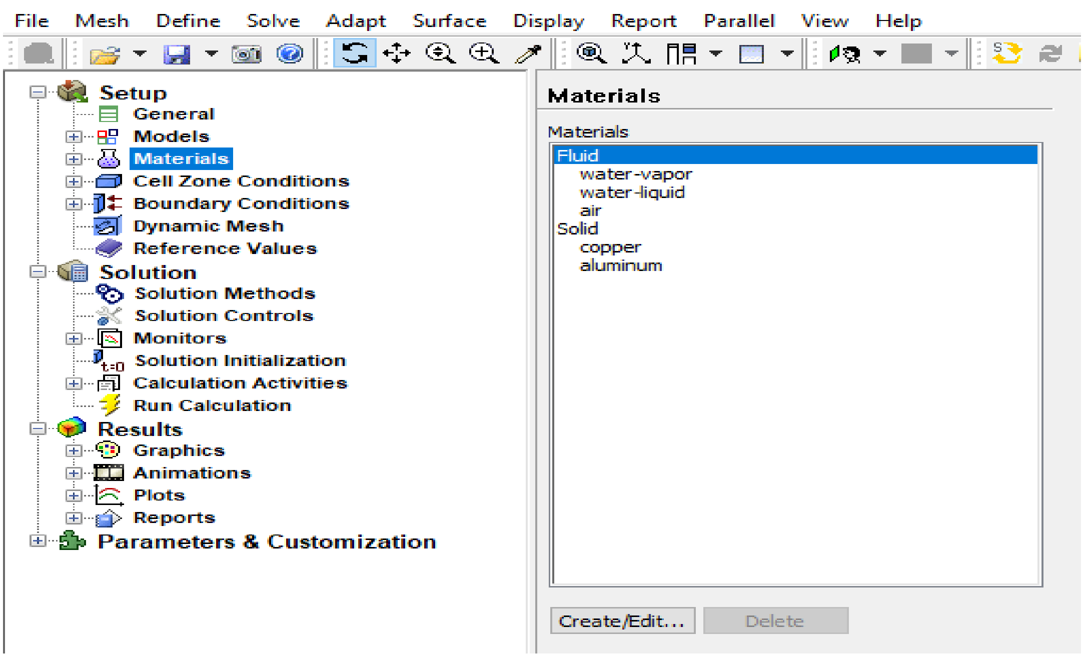


Fig 3.5 Opening ANSYS materials database

After clicking on Fluent Data base, the materials can be copied, and then they will be displayed on the first window. The materials selected here are only the fluids (Water liquid and water vapor), since aluminum is already default for the solid parts as well as air is for fluids.

3.7.2.4 MESH GENERATION

The Meshing tool can be invoked by right clicking on ‘Mesh’ at the workbench project and selecting ‘Edit’, similar to ANSYS-Design Modeller. The home interface of ANSYS Meshing tool.

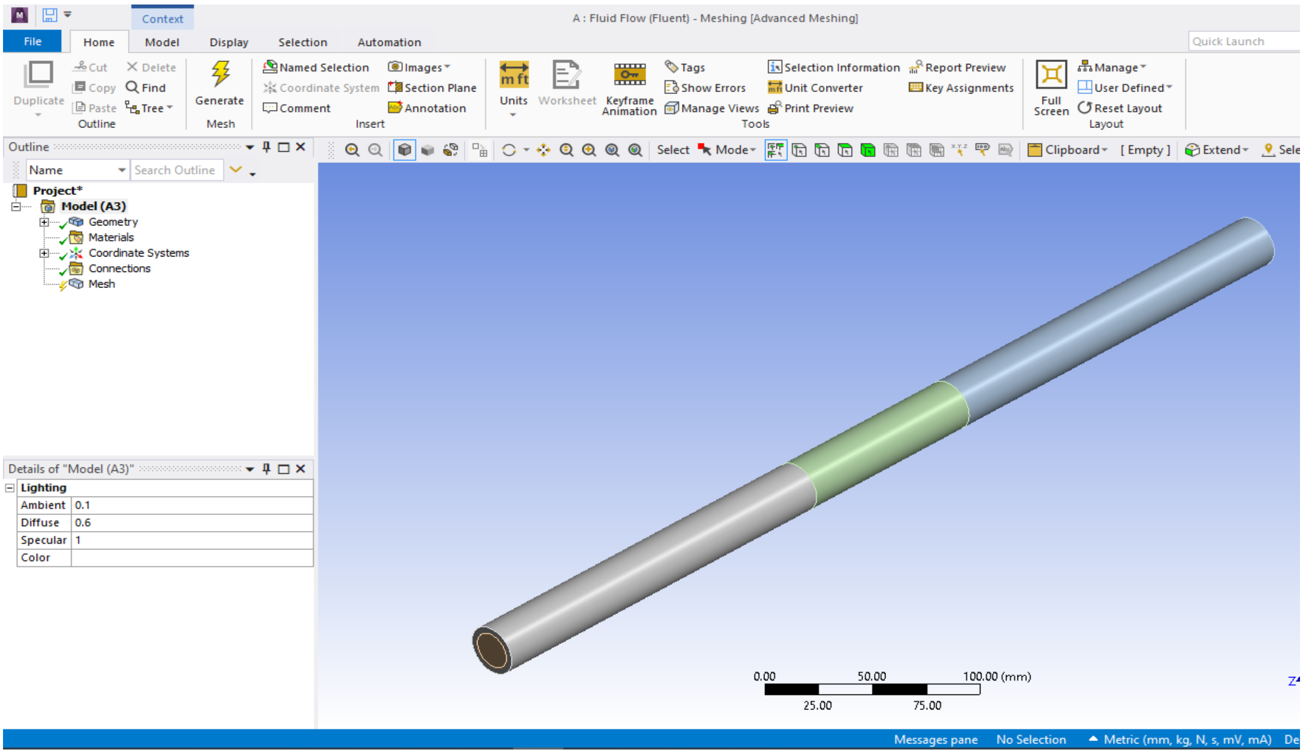


Fig 3.6 ANSYS Mesh Interface

For creating the mesh, the ‘Objects’ have to be identified. An object is generally a set of face zones and edge zones. Objects are generally closed solid volumes or closed fluid volumes. Different types of mesh can be created in ANSYS Meshing tool through ‘Insert’ option. The mesh settings can be adjusted according to the model requirement. After creating the mesh, the proceeds with assigning the ‘named selections’ for different parts of the model structure. This is helpful for defining the boundary conditions. The ‘named selections’ can be created by right clicking on the region and selecting ‘Create Named Selection’.

Figure 3.7 shows the sample of various regions that have been defined using ‘named selections. The boundary conditions have to be specified in the ANSYS Fluent for all these regions. Finally, the mesh can be generated by selecting ‘Generate Mesh’.

There are various mesh metrics to check the quality of the mesh. When the mesh is successfully developed, the ANSYS workbench is updated with the green tick mark.

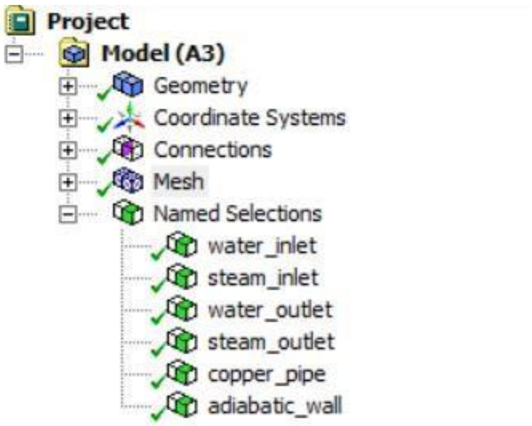


Fig 3.7 Named Selections

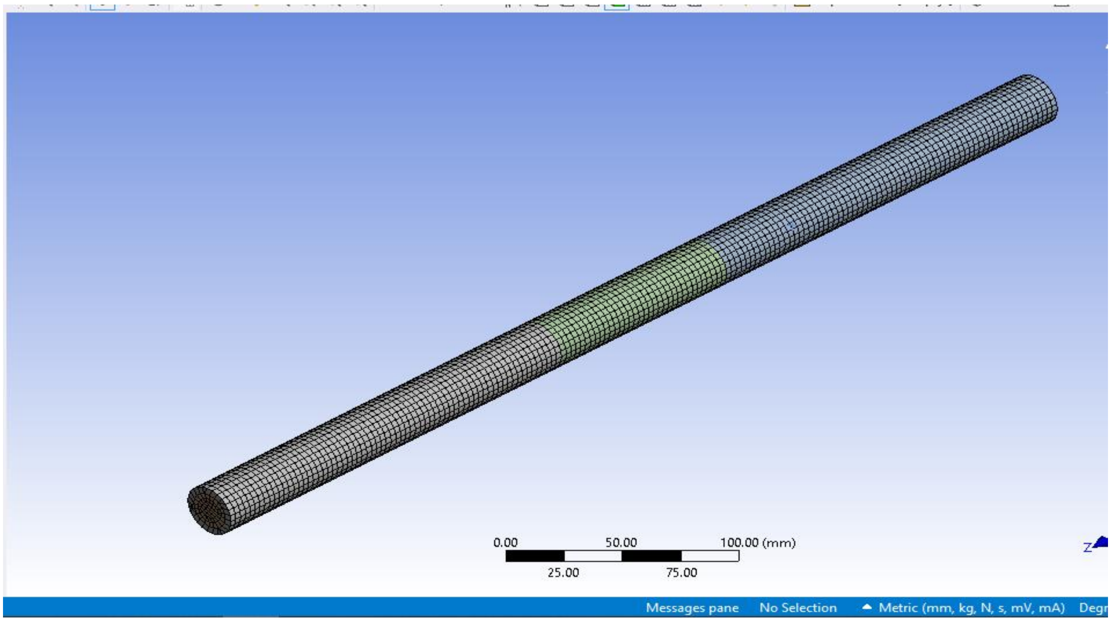


Fig 3.8 Fully Developed Mesh

3.7.2.5 DEFINING THE PHASES AND THEIR INTERACTIONS

Now that the materials were imported, come back to the Models, and then following the steps shown in the image 10, define which material corresponds to each of the three phases.

The phases are defined as

1- Primary phase Water vapor

2- Secondary phase Water liquid

3- Secondary Phase Air

The interactions between the phases need to be defined. Clicking twice on phase interactions, there are only two possible interactions: Mass transfer and surface tension. Both play an important role in a HP. First, the mass transfer accounts the effects of condensation and evaporation happening inside the pipe, second the surface tension is important because the tube diameter is very small and the liquid ends up sticking to the walls (as expected). To set up the mass transfer Then, clicking on edit, the saturation temperature, which is the temperature that the liquid starts to turn to vapor and the vapor to liquid, is defined as 35 °C (308 K) at a pressure of Pa.

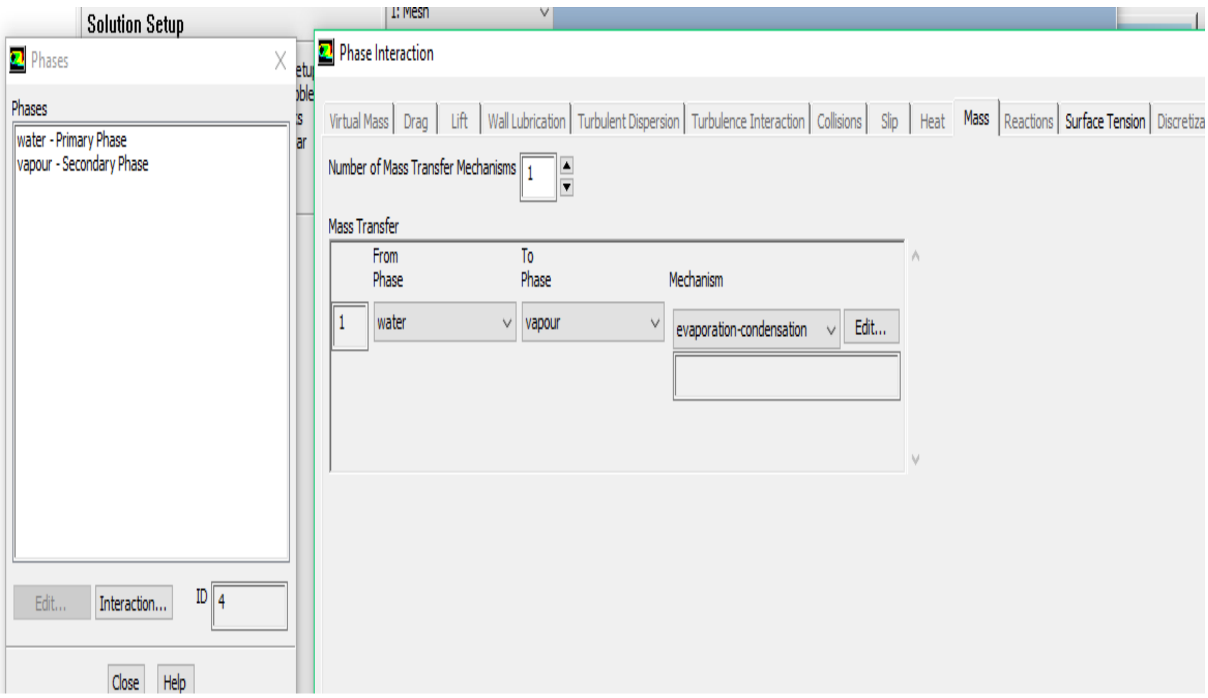


Fig 3.9 Setting up the mass transfer mechanism

To set up the surface tension, even though the surface tension changes with temperature, here it is considered constant (0.07 N/m), for simplification purposes, which is approximately the surface tension of water at 39 °C. Wall adhesion must be included since the main feature of a HP is the fact that the water sticks to the walls.

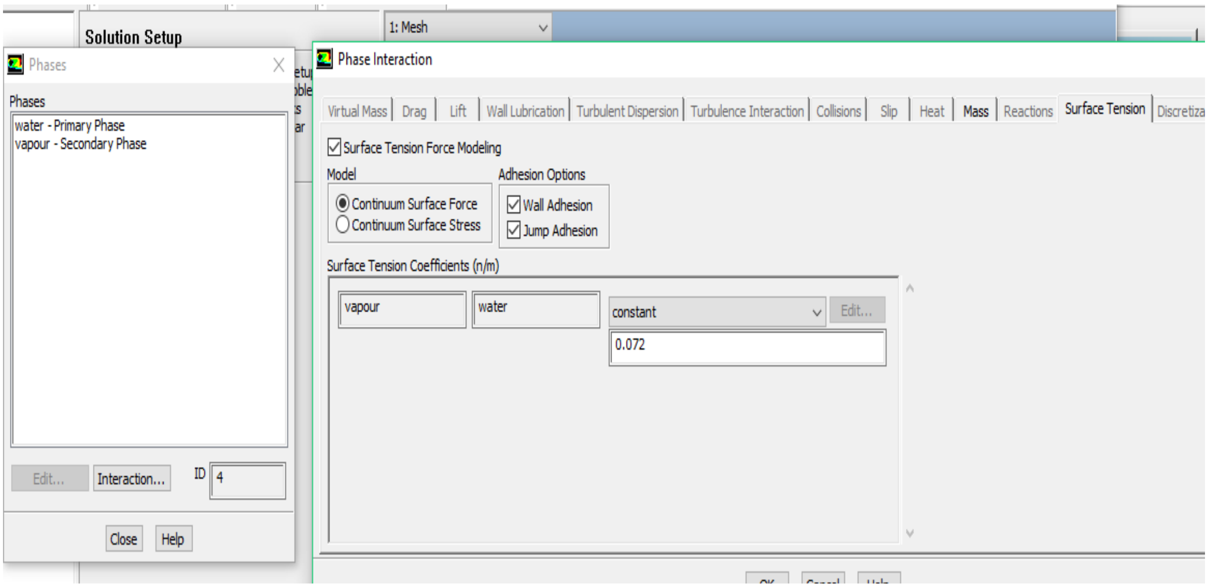


Fig 3.10 Setting up the surface tension

3.7.2.6 CELL ZONE CONDITIONS

On this section the vacuum inside the pipe is introduced. Clicking on “Cell Zone Conditions” and then on “Operating Conditions” set the “Operating Pressure” as 101325 Pa (this is the saturation pressure of the water at a saturation temperature of 29 °C). The other parameters used are the default ones.

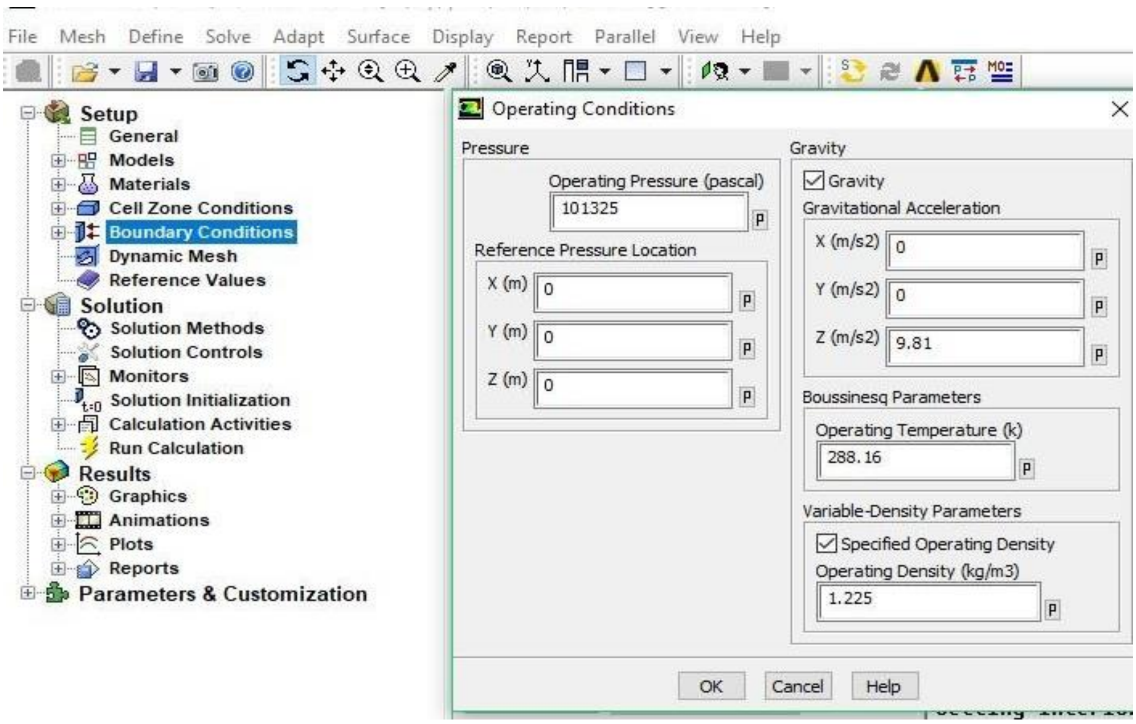


Fig 3.11 Including the vacuum effect

3.7.2.7 BOUNDARY CONDITIONS

The boundary conditions are introduced to ANSYS. Named selections should be created during the meshing to facilitate this step. After clicking on “Boundary Conditions” a window pops up. The following should be given as inputs.

Outer evaporator wall: constant heat flux

Outer condenser wall: convection

Adiabatic section: zero heat flux

Table 3.1

parameter	Value
Total length	0.5m
Evaporator length	0.2m
Adiabatic section length	0.1m
Condenser length	0.2m
Wick thickness	2mm
Wall thickness	4mm
Inner core radius	9.5mm
Saturated temperature	373.15k
Water density	998 kg/m^3
Vapor density	1.25 kg/m^3
Operating Pressure	101325 Pa

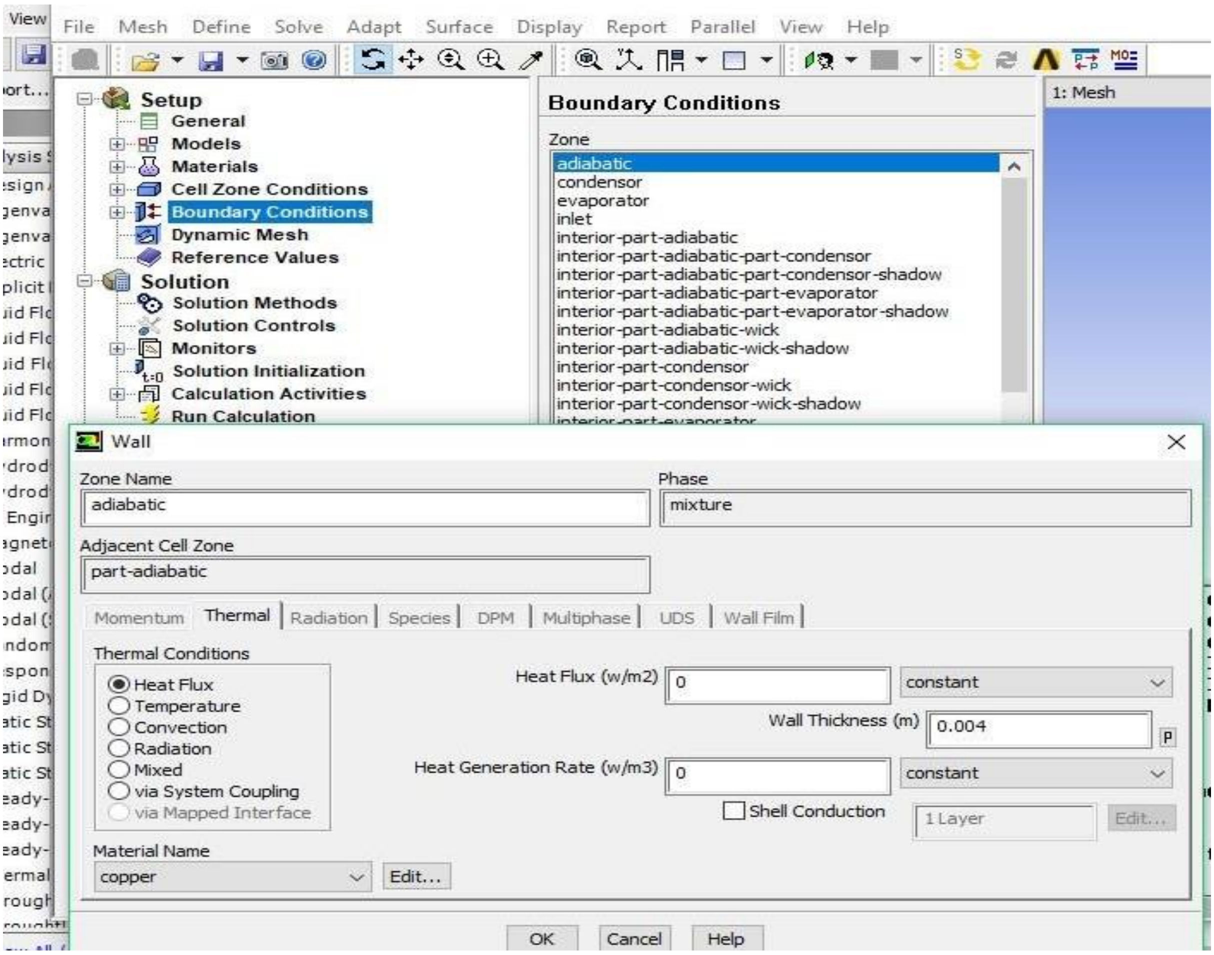


Fig 3.12 Boundary conditions for adiabatic section

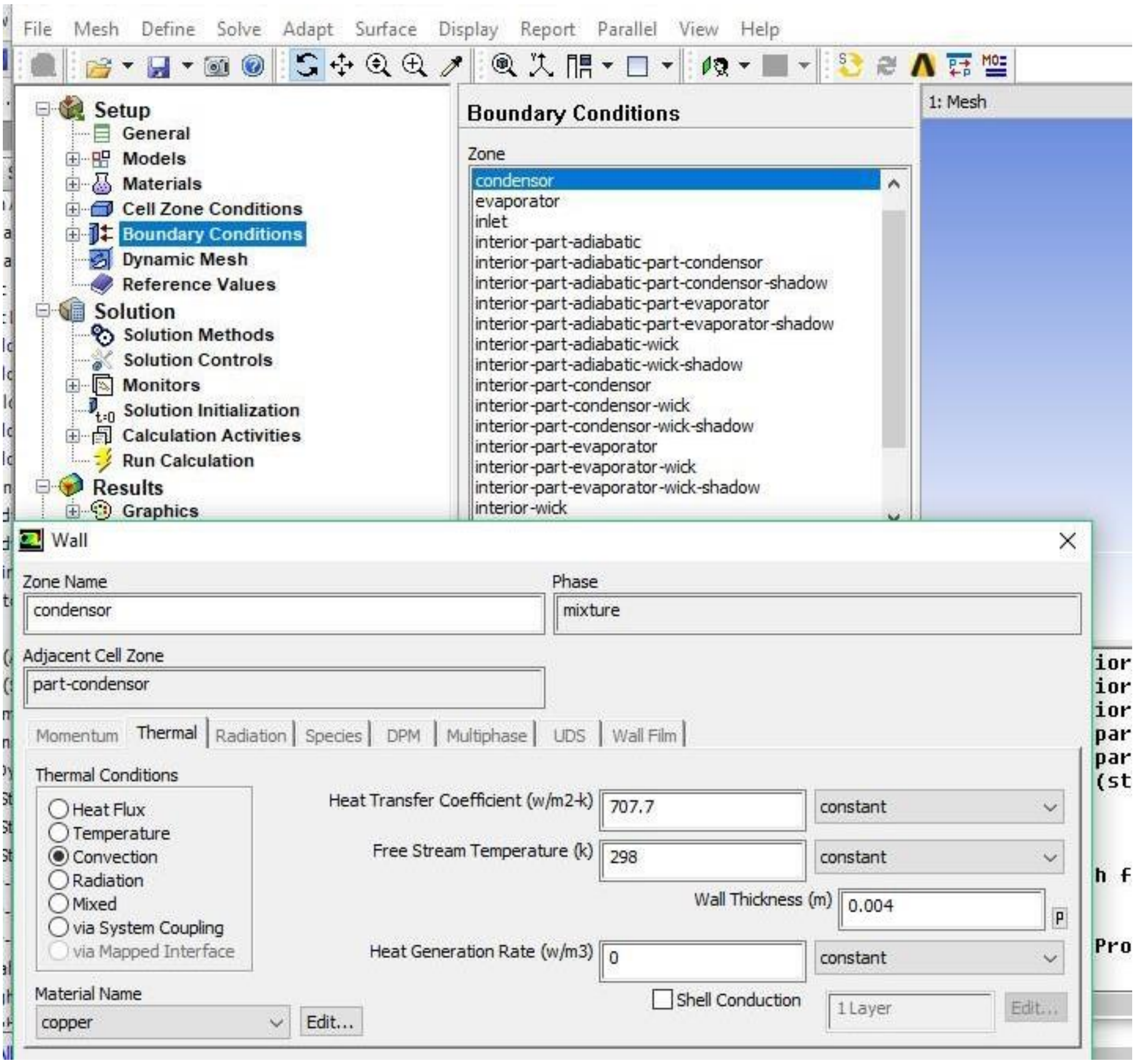


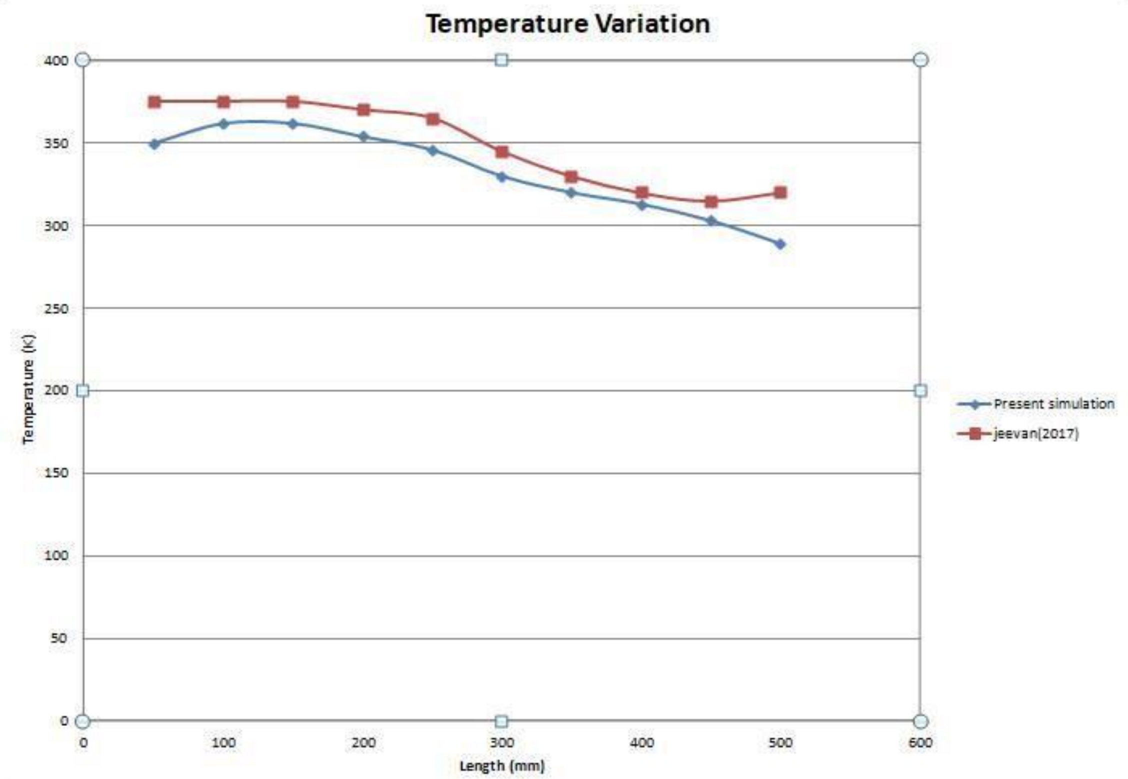
Fig 3.13 Boundary conditions for the evaporator

Chapter-4

RESULTS AND DISCUSSIONS

In this work, we have first taken the parameters from a journal and simulated the same in ANSYS. Once our results are matched to the results from journal paper [18] we changed the various parameters. The heat flux at the evaporator section is changed to which resulted in the reduction of evaporation temperature of the fluid used. The simulation of heat pipe was conducted with and without the wick structure. The wickless heat pipe will work with the help of gravity which represents the gravity assisted heat pipe or a thermos-syphon. The heat pipe with wick material even work against the gravity which depicts the standard heat pipe. Different heat inputs, different pipe materials and two different working fluids which have varying the useful range of temperature are tested in this study.

4.1 VALIDATION



%error of validation = 4.7%

Fig 4.1 Variation of Temperature Comparison from Jeevan Paper [18]

4.2 VARIATION OF TEMPERATURE AT DIFFERENT HEAT INPUTS

Axial temperature profiles are drawn from the data of temperatures that is obtained at different axial distances on the heat pipe body. The axial temperature distribution along the heat pipe for is shown in Figure. Actually, the figure represents the evaporator, adiabatic section, and condenser temperature variations with distance.

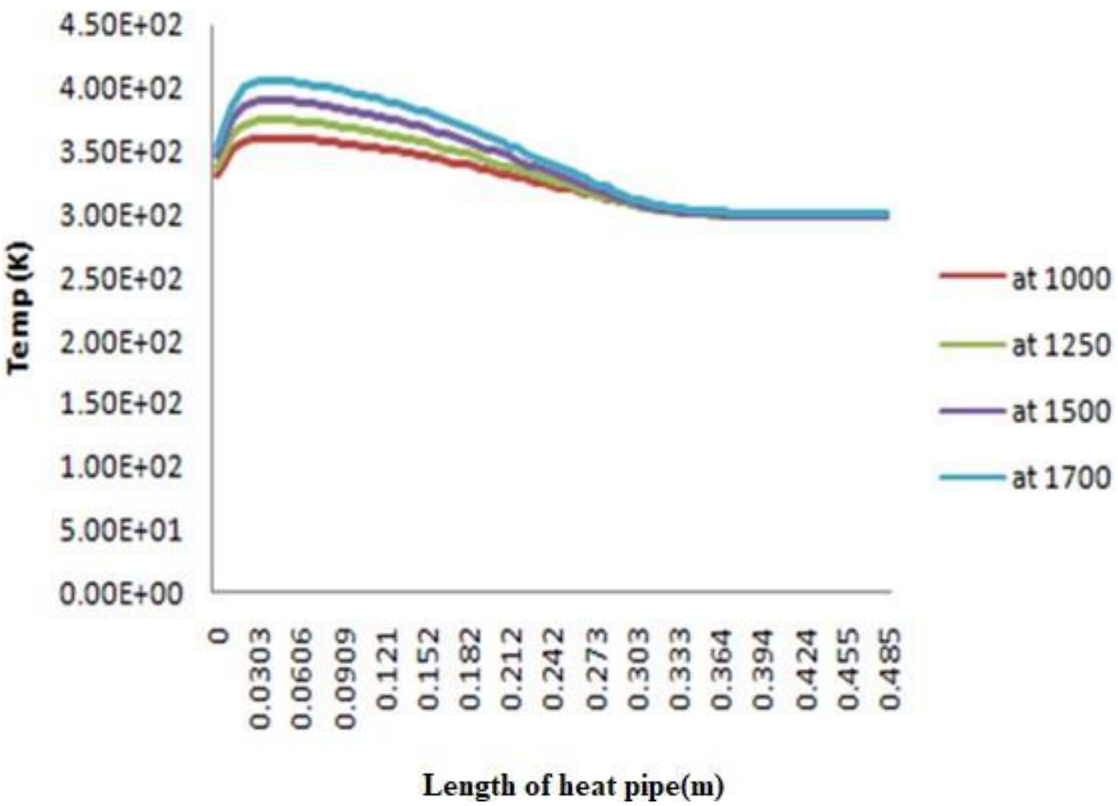


Fig4.2 Temperature variation at different heat inputs.

The figure shows that the gradient of axial temperature distribution increases with heat input and shows larger temperature differences across the condenser and evaporator section. The maximum temperature of the heat pipe was 402K, 390K, 370K and 360K at a heat flux of 1000W/m^2,1250W/m^2,1500W/m^2 and 1700W/m^2 respectively. The trend is obvious since greater temperature slope is required for increased heat transfer in case of simple conduction heat transfer.

4.3 TEMPERATURE PROFILE FOR DIFFERENT PIPE MATERIALS

The temperature distribution for different heat pipe shell materials are obtained along the length of the heat pipe body. The temperature profile is drawn at a heat flux of 1000W/m^2. The temperature of the aluminum is more at the evaporator section compared to that of the copper and stainless steel which implies that it can absorb more amount of heat from the source. At the condenser section the temperature of the stainless steel is low. The copper pipe will operate with a less temperature gradient will have more effective thermal conductivity. Thus the copper heat pipes will be the best choice for many applications. Different materials will have different operating temperatures which is shown in fig. the selection of the heat pipe will be based on the operating temperature and the compatibility of the working fluid with the pipe material. The aluminum material will not compatible with the water as the working fluid and similarly the ammonia fluid will not be compatible with copper as the pipe wall material and the wick structure.

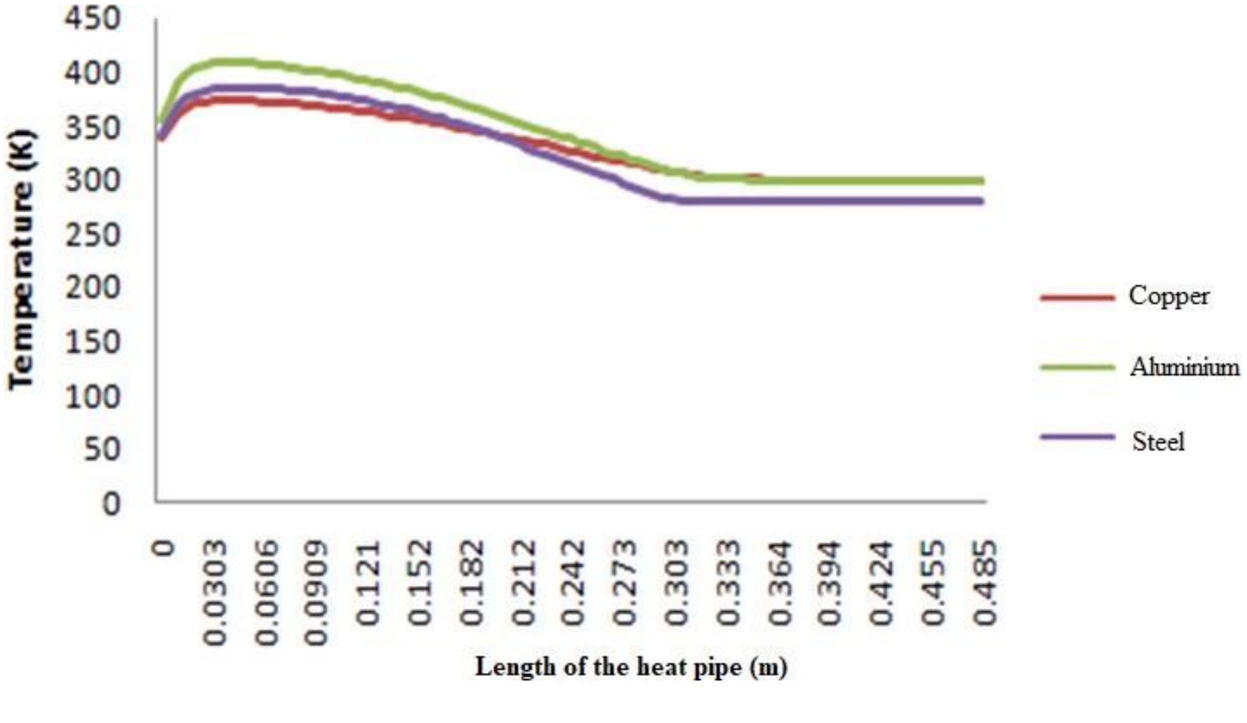


Fig 4.3 Temperature profile for different pipe materials

4.4 COMPARISON OF WICK AND WICKLESS HEAT PIPE:

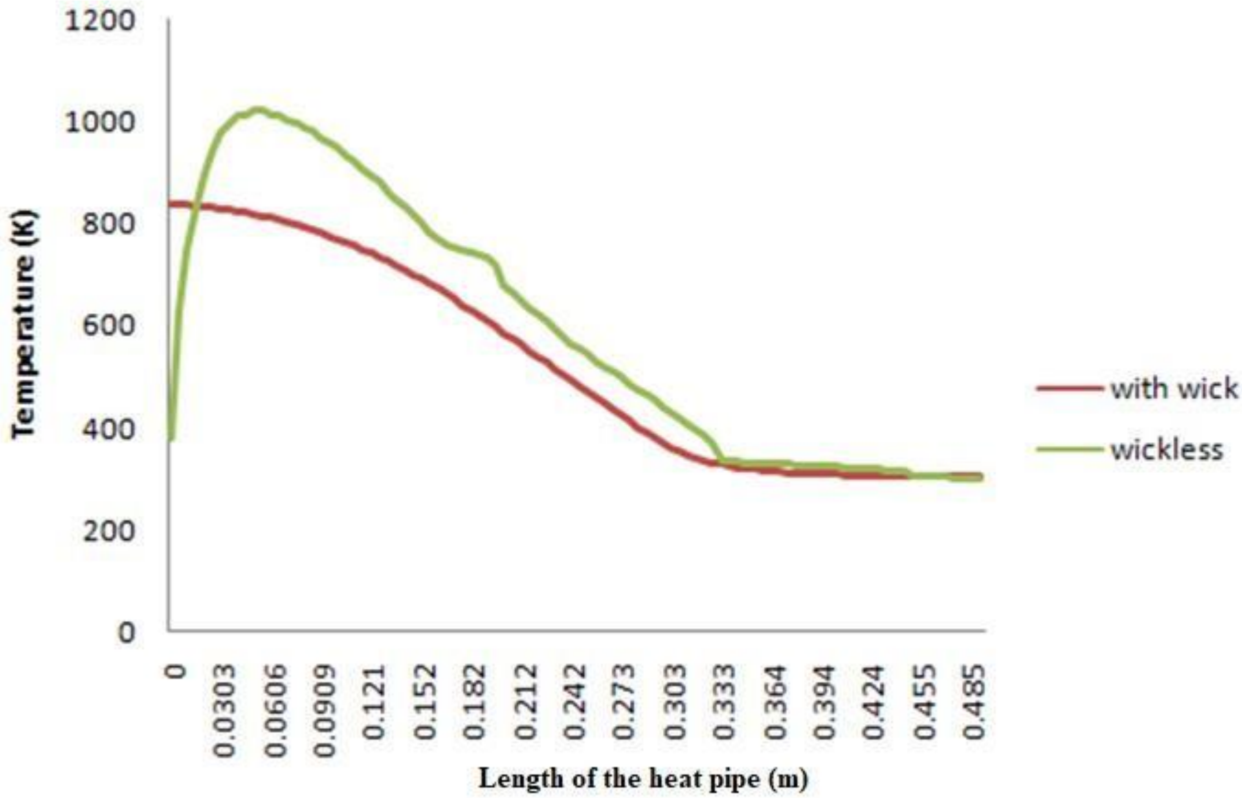


Fig 4.4 Wick vs Wickless heat pipe

The temperature distribution profile for a wick and wickless heat pipe was drawn along the length of the heat pipe at a heat flux of 10000W/m^2. The temperature of the wickless heat pipe was more compared to that of a wicked heat pipe because in a wicked heat pipe the fluid from the condenser section is return to the evaporator section with the help of the capillary wick structure and the gravity. But in a wickless heat pipe due to the absence of the capillary wick structure the fluid will return to the evaporator with the help of the gravity only. The operating temperature is in the range of 805K for a wicked heat pipe and 1002K for a wickless heat pipe. The capillary structure will transfer the fluid from the condenser to the evaporator this will ensure the continuous transfer of the fluid from the condenser section to the evaporator section. Due to continuous mass transfer of the working fluid more heat is transferred within the heat pipe and less temperature difference between the evaporator and condenser section. In the wickless heat pipe the condensate will return to the evaporator by the surface tension force which is acted in between the fluid and the wall.

4.5 EFFECT OF WORKING FLUID

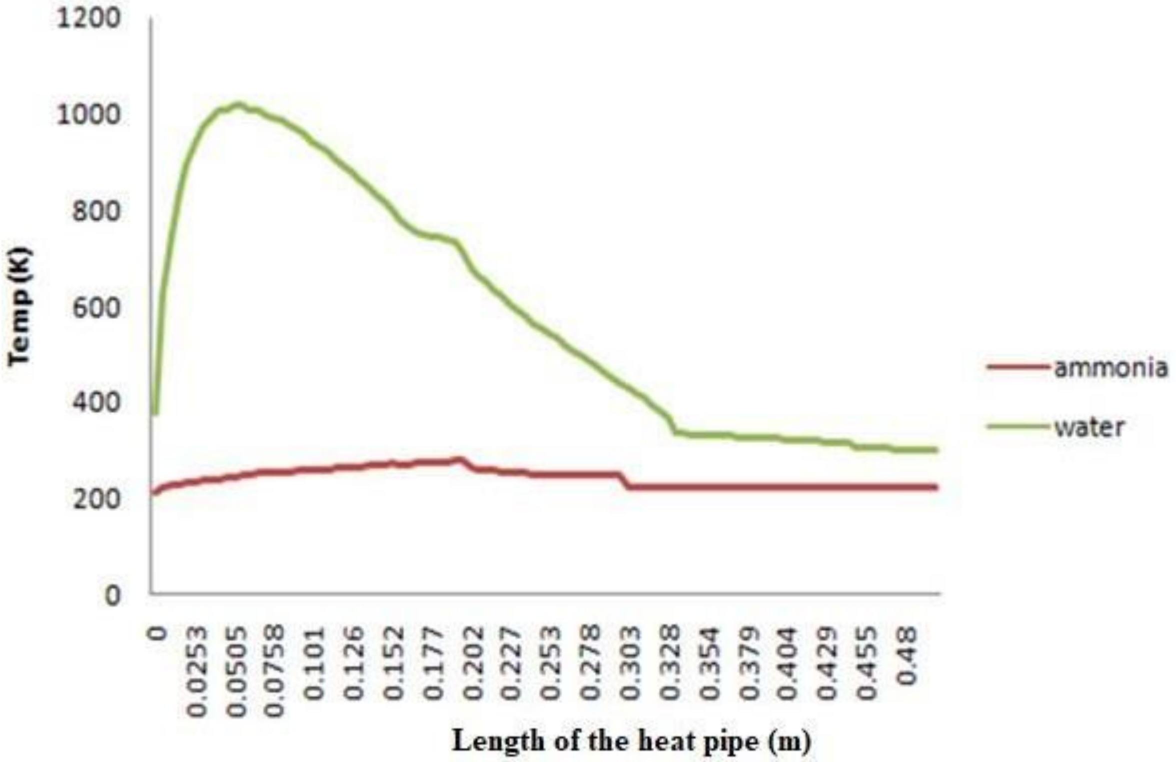


Fig 4.5 temperature variation for different working fluids

The temperature profile for water and ammonia as working fluid for the heat pipe is drawn at a heat flux of $10000 W/m^2$ at the evaporator section and a convection heat transfer coefficient of $707.7 W/m^2$-k with a free stream temperature of 301.45K for water and 210K for ammonia fluid. For the same operating conditions, the temperature profile for the water is above that of ammonia which indicates that water will work at high operating temperatures compared to that of ammonia. At low operating conditions i.e., below 0^0C water will become freeze and the heat pipe will not operate and tends to be failed. For that condition ammonia heat pipes will be considered. The temperature gradient for ammonia heat pipes is low compared to that of water, due to low temperature gradient the heat transfer rate is high for ammonia heat pipes and they will work at uniform temperature range which will reduce the thermal resistance of the network.

4.6 MINIMUM AND MAXIMUM TEMPERATURE RANGE

4.6.1 FOR DIFFERENT HEAT INPUTS

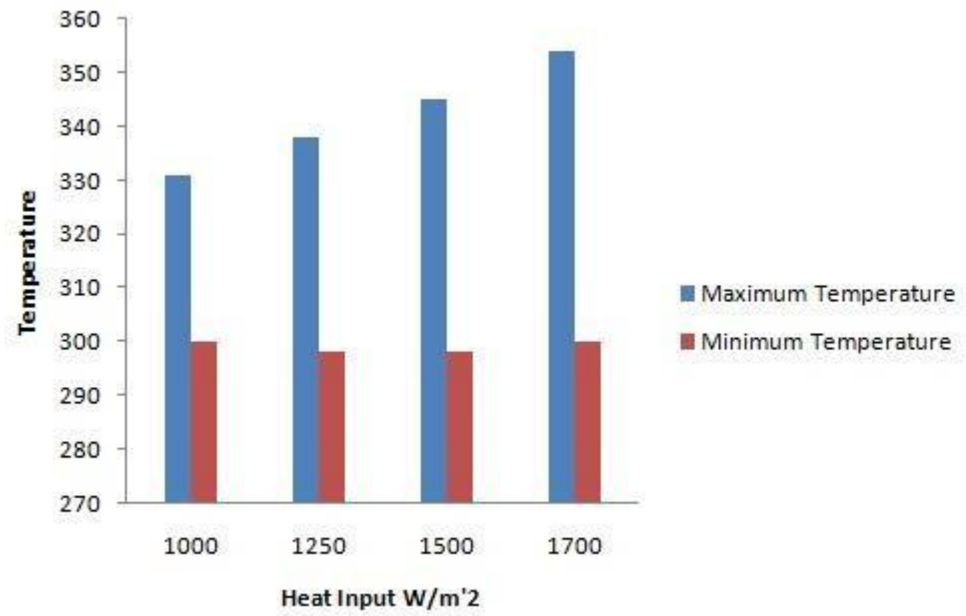


Fig 4.6 Minimum vs maximum temperature at different heat inputs

The minimum and maximum temperature range at different heat fluxes was obtained and the results are plotted with water as the working fluid. At different heat fluxes of 1000W/m^2,1250W/m^2,1500W/m^2 and 1700W/m^2 the maximum temperature was 330K,338K,346K,358K and the minimum temperature was 300K,298K,298K,300K respectively. The slope of the temperature profile increases gradually with increase in the heat flux. The maximum temperature for the heat pipe is high at a heat flux of 1700W/m^2 which indicates that the increase in heat flux will increase the heat transfer by reaching the boiling point of the working fluid. At the condenser section there is a uniform temperature of the working fluid although there is change in heat flux at the evaporator which shows that the working fluid will condensate at the condenser section before it enters into the evaporator. It will act as a perfect heat sink to transfer the heat from the whole network.

4.6.2. FOR DIFFERENT WORKING FLUIDS

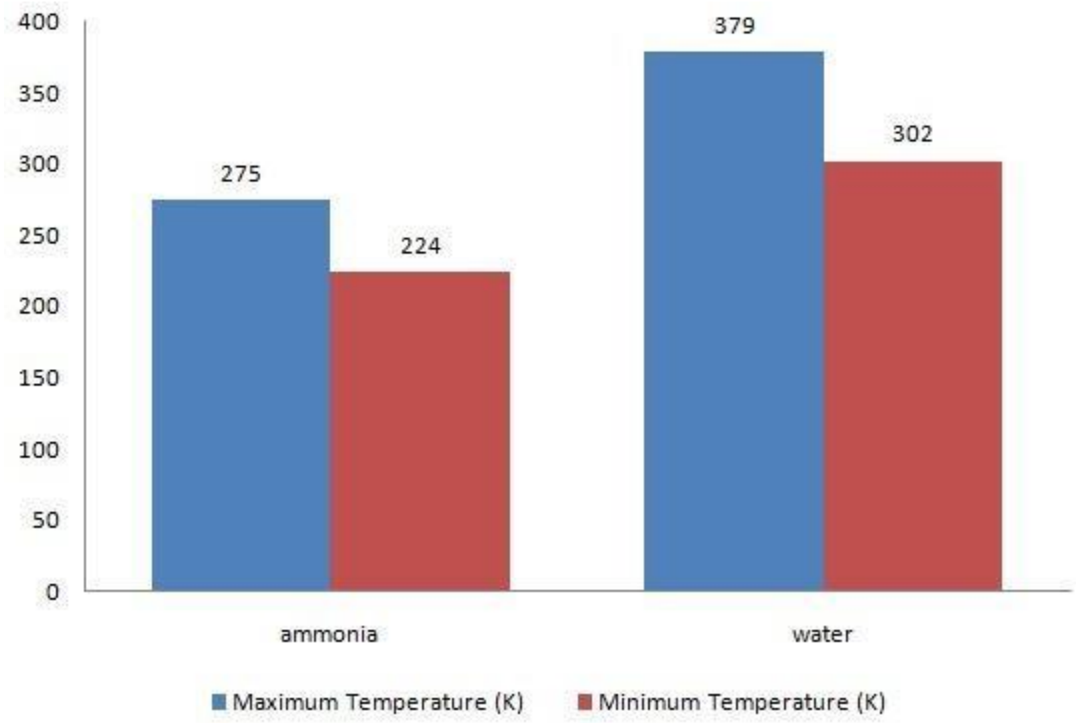


Fig 4.7 Minimum and maximum temperatures of working fluids

The minimum and maximum temperature for ammonia and water as the working fluids is shown in the above figure at a heat flux of $10000 W/m^2$. The maximum operating temperature range is 270K for ammonia heat pipe and 373K for water heat pipe. The free stream temperature or the room temperature is 220K for ammonia and 301K for water. This minimum and maximum temperature will provide the operating range of the working fluid for a particular temperature conditions.

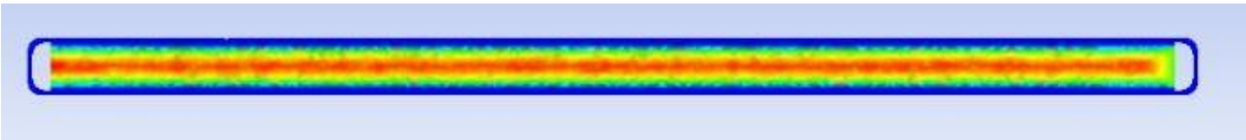


Fig 4.8: Velocity contour

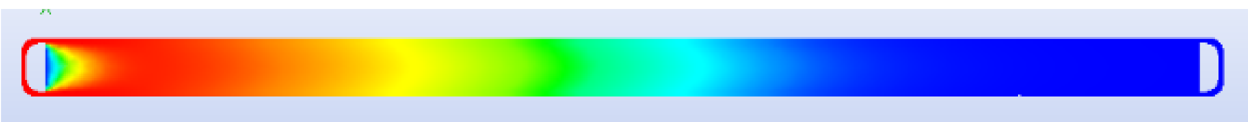


Fig 4.9: Temperature contour

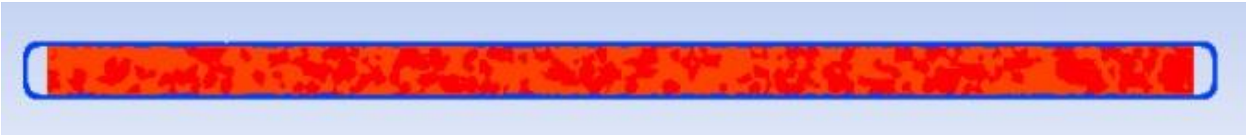


Fig 4.10: Pressure contour

4.7 THERMAL RESISTANCE

The effectiveness of a heat pipe is determined indirectly in terms of the thermal resistance of the heat pipe. The heat transfer rate of any system is indirectly proportional to its thermal resistance. The system with low thermal resistance will transfer more amount of heat. Hence the effectiveness of a system is high with the low thermal resistance.

4.7.1 THERMAL RESISTANCE OF A MATERIAL

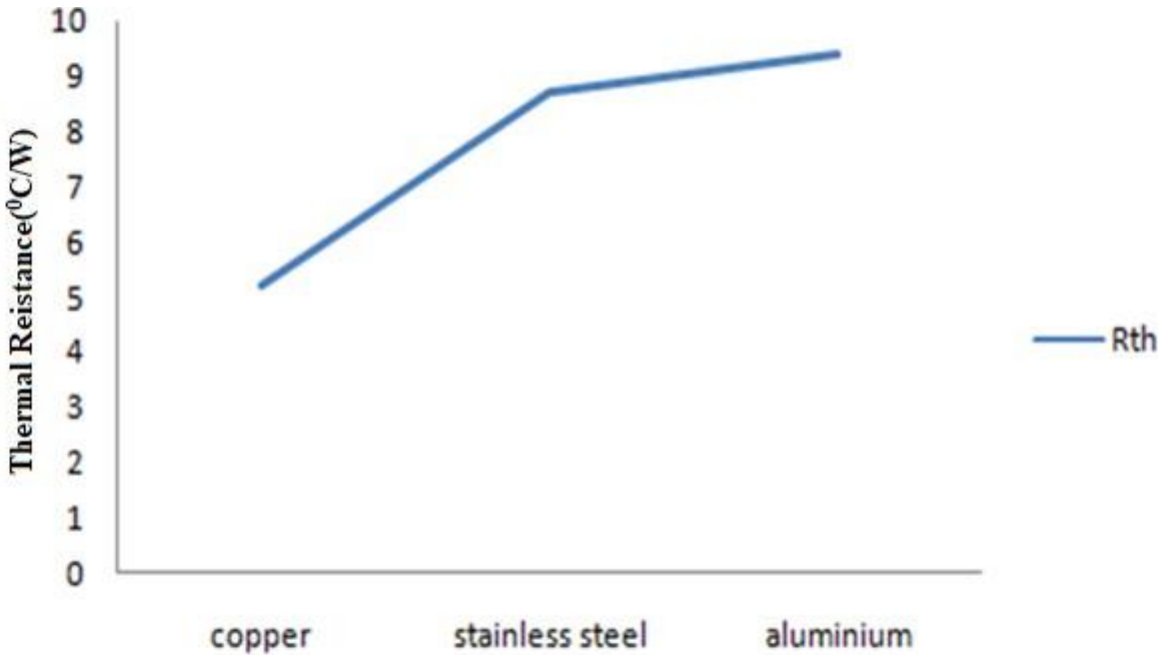


Fig 4.11 Thermal resistance Vs Materials

The thermal resistance of different shell materials is compared at a heat flux of 1000W/m^2 which is shown in the above figure. Three materials such as copper, stainless steel and aluminum are taken into consideration for comparing the effective thermal resistance of the network. The thermal resistances are calculated by considering the average temperatures of both the evaporator and the condenser. The thermal resistances of the materials are 5.196K/W, 8.71K/W and 9.38K/W for copper, stainless steel and aluminum respectively. The effective thermal resistance of the copper is low compared to that of the remaining materials. So the heat transfer rate of copper pipe is more of the remaining materials and it has high thermal resistance.

4.7.2. VARIATION OF THERMAL RESISTANCE WITH HEAT FLUX

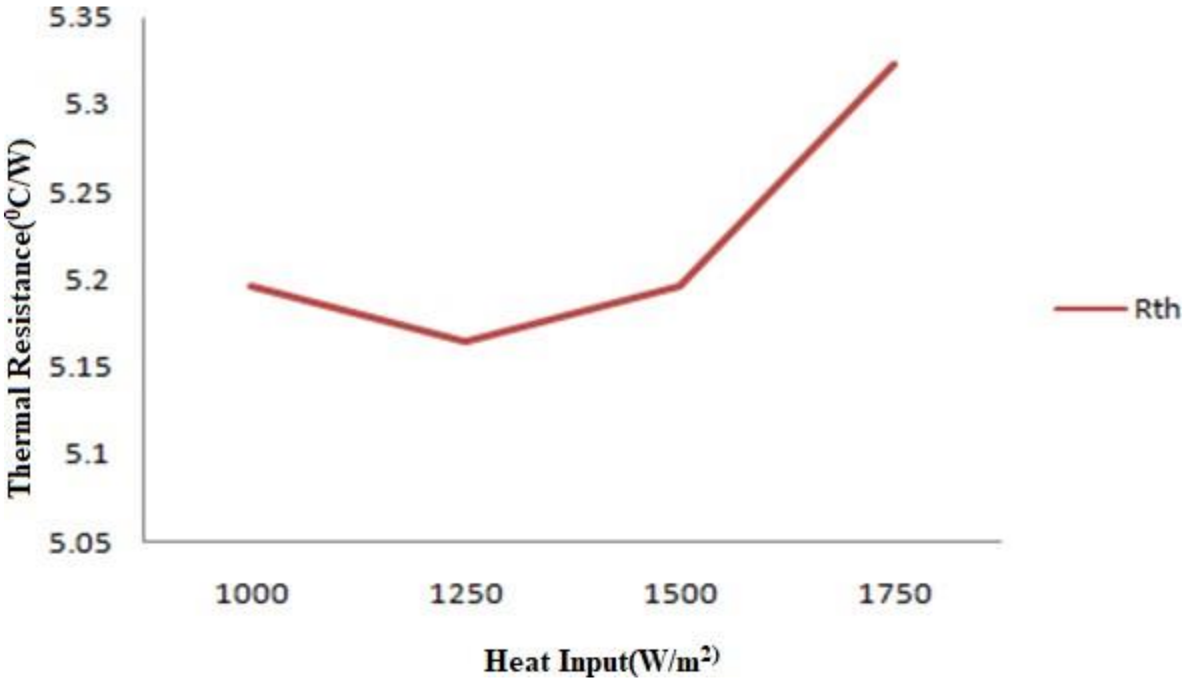


Fig 4.12 Thermal resistance Vs Heat flux

The variation of thermal resistance by changing the heat flux was calculated which is shown in the figure. The thermal resistance was varied from 5.16-5.32K/W by changing the heat flux from 1000-1750W/m^2. Initially with increase in heat flux there is a gradual decrease in thermal resistance up to 5.196K/W at a heat flux of 1250W/m^2. But on further increasing the heat flux the slope will increase linearly. This sudden change in resistance will be small which will not affect the performance of the heat pipe since the change in resistance is negligible.

4.7.3 VARIATION OF HEAT TRANSFER COEFFICIENT WITH HEAT FLUX

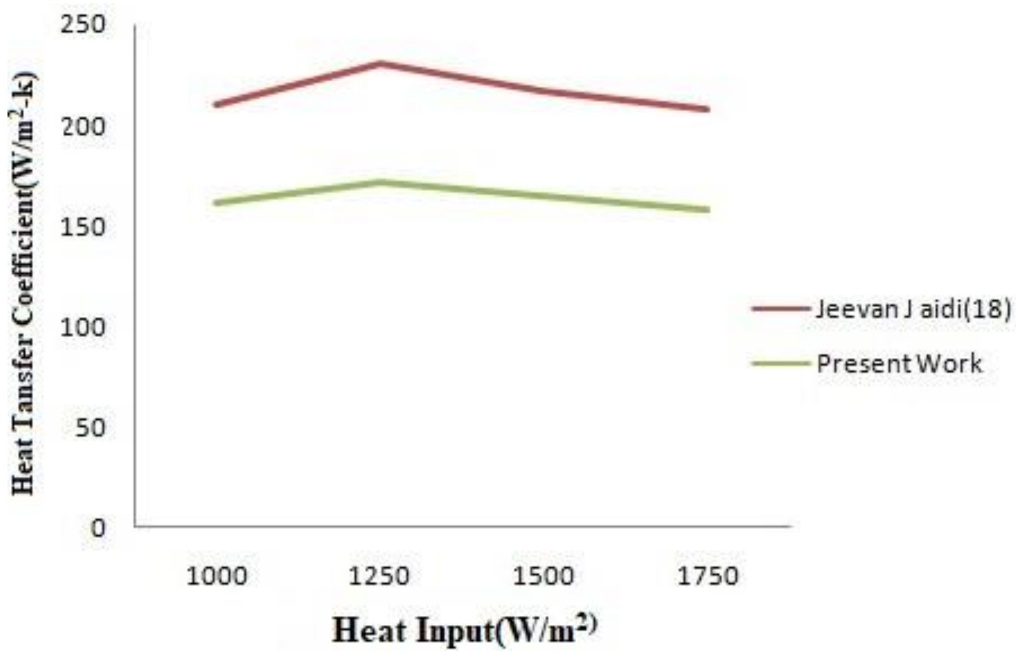


Fig 4.13 Heat transfer coefficient Vs heat flux

The variation of heat transfer coefficient by changing the heat flux is determined and the same is compared with the previous experimental data. The maximum heat transfer coefficient is 170W/m^2-k at a heat flux of 1250W/m^2 but the experimental value is 238W/m^2-k.there is a small deviation of the computational value from the experimental value due to improper boundary conditions and due to the consideration of assumptions such as heat transfer in 1-direction.

4.8 DISCUSSION

From the above graphs, we have seen trends in variation of different factors with the given heat input. From this, we can suggest the material, liquid to be used for various applications depending on the temperature difference between the evaporation section and condensation section. The temperature difference is one of the main factor in selecting the material of heat pipe and the liquid to be used in the heat pipe, as different liquids have different properties.

4.9 CONVERGENCE

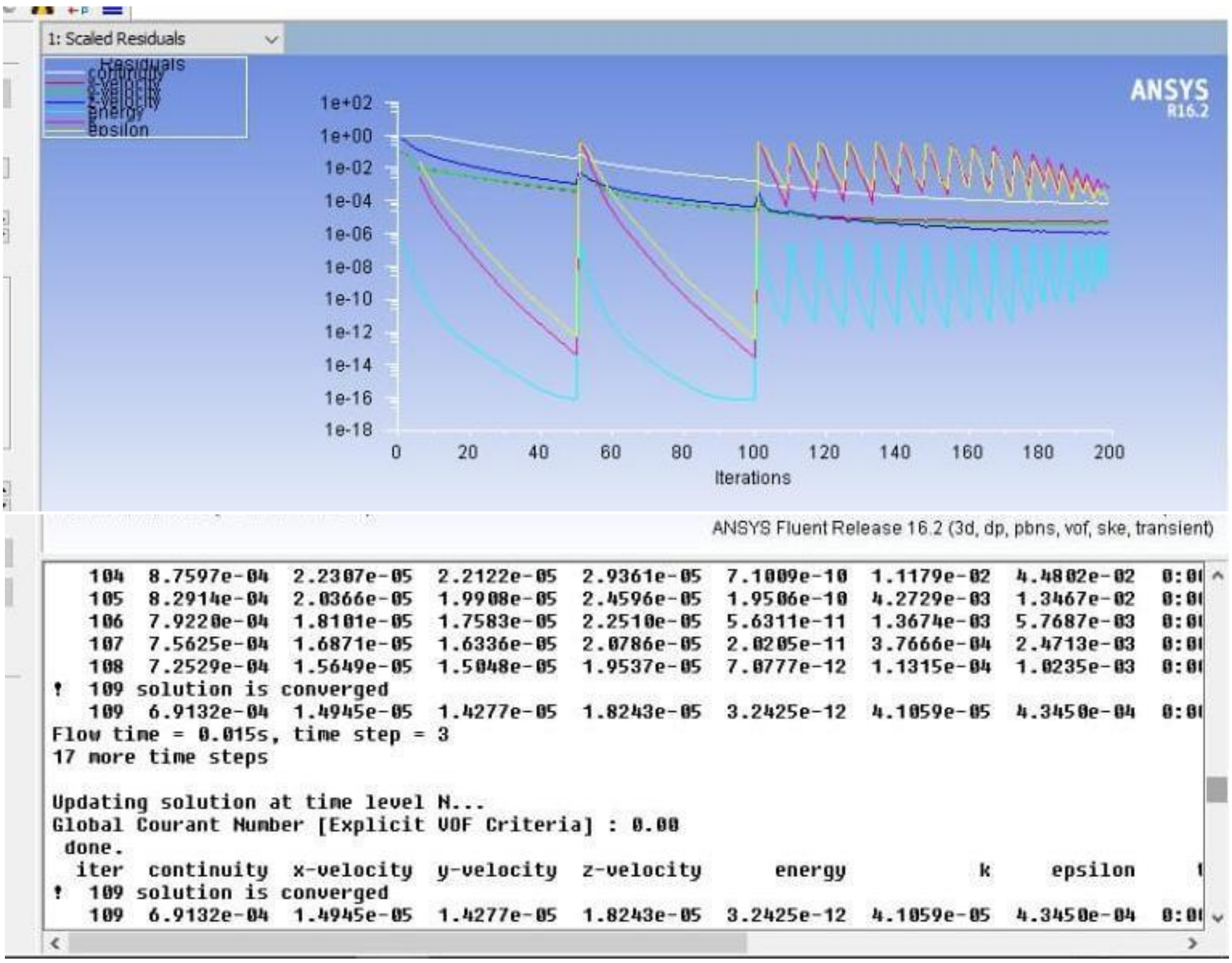


Fig 4.14 Numerical Representation Convergence

In problems with non-unique solution it is normal to find different solutions when some numerical parameters are changed, in this case the convergence criteria is applicable. There is no correct value or parameters, if it converges to a solution all solutions are equally valid.

The final solution is converged.

Chapter-5

CONCLUSION

In this work, Thermal flow analysis is conducted on a heat pipe by varying the pipe material and the working fluid and comparison is done between the results obtained, the conclusions that we have obtained from the previous chapter are

- The Heat Pipe evaporates the fluid at higher temperature at higher heat flux inputs compared to lower heat flux.
- The temperature in the condenser section of the Heat Pipe won't change significantly with the change in heat flux.
- Copper is effective compared to that of Stainless steel and Aluminium because of the less thermal resistance offered by the copper.
- The wicked Heat Pipe is better compared to wickless Heat Pipe because of the continuous mass transfer due to capillary action in the wick structure.
- Heat pipe works significantly at lower temperature when taken ammonia as working fluid compared to water which works significantly at higher operating temperature.

REFERENCES

[1] Aloke Kumar Mozumder, and Abul Fahad Aakon, (2011), *Characteristics of Heat Transfer For Heat Pipe And Its Correlation* ,Vol.2011 Article Id 825073

[2] K snehith and S. Bhanu Prakash, (2019), *Design of Heat Pipe Based on Capillary and Entrainment Limitations*, Vol.2200,020069

[3] Leonaed L. Vasiliev, (2003), *Heat pipe in modern heat Exchangers*, Applied Thermal Engineering: Vol.25(2005) 1-19

[4] Xiaoming Huang, Chunyu Shia, (2019), *Performance analysis and Design Optimization of heat pipe sink with a variable Height Fin Array under Natural Convection*, Vol.159,113939

[5] H Jouhara, A Chauhan, T Nannou, (2017), *Heat pipe based systems-advances and applications*, Energy 128729-754

[6] Aditya Narayan, Siddharth parailkar and Sameer Khandekar, 2015, *Development and Testing of Flexible heat pipe*, paper no.IHMTC 2015-781 17-20

[7] Stephane Lipsa, Valerie Sartrea, Frederi Lefevre 2016 , *Overview of heat pipe studies during the period 2010-2015*, interfacial phenomena and heat transfer,4(1),33-53

[8] Huei Chu Weng and Meng-mHsueh Yang 2018, *Heat transfer performance enhancement of gravity heat pipes by growing AAO nanotubes on inner wall surface*, Inventions 3030042,3,42

[9] B.Borgmeyer, P.Cheng and Y.Zhang, 2017, *A Mathematical Model Predicting Heat Transfer Performance in a Oscillating heat pipe*, IPACK2007-33246 812

[10] Daehoon Lee and Chan Byon 2018, *Fabrication and characterization of pure-metal-based submillimeter-thick flexible flat heat pipe with innovative wick structures*, Vol 122,306-314

[11] Davoud Jafari,Wessel W Wits,Bernard J Geurts, (2017), *An investigation of porous structure characteristics of heat pipes made by additive manufacturing* ,THERMINIC 2017

[12] Shen-Chun Wu,Chuo-Jeng Huang,Sheng-Hao Chen (2015), *Investigating the effect of powder-mixing parameter in bi-porous wick manufacturing on enhancement of loop heat pipe performance,* Vol-89,460-467

[13] NamponSangpab,Nobuhiro Kimura (2019), *Combined effect of bending and flattering on heat transfer performance of cryogenic sintered-wick heat pipe,* Vol-148,878-885

[14] Hai Tang, LixianLian, Jing Zhang,Ying Liu (2019), *Heat transfer performance of cylindrical heat pipes with axially graded wick at anti-gravity orientations*, Vol-163,114413

[15] FeiXin,Ting Ma,Qiuwang Wang (2018), *Thermal performance analysis of flat heat pipe with grade mini grooves wick,* Vol-228,2129-2139

[16] Ixue Zhang, Junyu Liu, (2019), *Numerical simulation and analysis of heat leakage reduction in loop heat pipe with carbon fiber capillary wick,* Vol-146,106100

[17] Y.Wang,K.Yafai, (2000), *An Experimental Investigation of the Thermal Performance of an Asymmetrical Flat plat heat pipe* , Vol.43(2000) 2657-2668

[18] Abhishek Gaikwad, Jeevan jaidi, (2017), *CFD Analysis of Wickless Heat pipe*, VOL IHMTC2017-09-0865.