INTERNATIONAL JOURNAL OF RESEARCH IN MECHANICAL, MECHATRONICS AND AUTOMOBILE ENGINEERING (IJRMMAE)
ISSN: 2454-1435 (Print) | 2454-1443 (online)
Volume 2 Issue 5 – www.ijrmmae.in – Pages 111-120

ANALYTICAL AND NUMERICAL ANALYSIS OF THERMOSYPHON

S.K.Narendranathan[1]

Assistant Professor, Department of Mechanical Engineering, Agni College of Technology, Chennai.

Abstract- Over the years, several improvements are being made in better energy saving methods. Heat pipes are being exclusively used in industries as a heat exchanger and due to its increased usage, there comes a need as well to improve its thermal efficiency. The multiphase characteristics of heat pipes make the analysis process tedious and complicate which results in lesser number of studies made on it so far. This paper describes a CFD model of a thermosyphon, a type of heat exchanger and to analyze its efficiency by varying its fill volume. A numerical model of thermosyphon in 'ANSYS Fluent' is developed.

A two-dimensional CFD model has been developed to visualize the evaporation and condensation processes taking place in the thermosyphon. Water is selected as the working fluid and copper is chosen as the thermosyphon material. Temperature profile of the thermosyphon is carefully examined to predict the efficiency of the process. In first case, the evaporative region of the thermosyphon is filled with 50% of the working fluid. For the second and third, the evaporative section is filled with 30% and 70% respectively. The complex processes of evaporation and condensation with the complex phase changing phenomenon in the thermosyphon is clearly visualized in ANSYS Fluent from which the performance can be easily studied.

Keywords— Thermosyphon, Energy saving, Temperature.

1 Introduction

Heat pipe is a device that has very high thermal conductance. The idea of heat pipe was first suggested by Gaugler in 1942 and the device was invented in the year 1960s by Grover. The vital feature of a heat pipe is its wicked structure which helps in the circulation of fluid within the system by evaporation and condensation against the gravitational force. The earlier devices like Perkins tubes which were used much before the invention of heat pipes, were not provided with wicks.

It was a closed tube that used water as a working fluid. The working fluid can be either in single or two-phase cycle for transferring heat to the boiler from the furnace. A typical Perkins tube used in furnace is illustrated in the Figure 1. Here, the tube circulates the fluid between the furnace and boiler.

INTERNATIONAL JOURNAL OF RESEARCH IN MECHANICAL, MECHATRONICS AND AUTOMOBILE ENGINEERING (IJRMMAE)
ISSN: 2454-1435 (Print) | 2454-1443 (online)
Volume 2 Issue 5 – www.ijrmmae.in – Pages 111-120

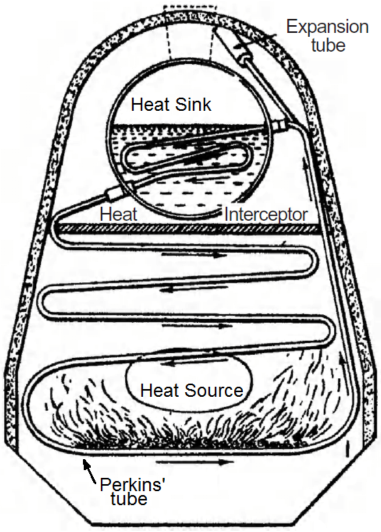

Figure 1: Perkins' tube

In a wickless heat pipe, heat is usually added to the evaporator section located at the bottom. This is where the working fluid is held initially. Then, the vaporized fluid from the evaporator section passes through the adiabatic section and reaches the condensation section. Here, phase change occurs as the working fluid loses its latent heat. The condensed fluid again returns to the evaporator section by means of gravity.

The increased need for energy conversation resulted in the commercial applications of heat pipes. The heat pipes are simple in design, efficient, flexible and compact. Because of this, they have been extensively used in wide range of applications like heating, ventilation and air conditioning systems, water heating systems and thermal management systems.

The usage of heat pipes increases in the field of technology and so studies are being made for optimizing and improving the performance of the heat pipes in the recent times. But, the number of studies made so far on the wickless heat

pipes is infinitely small when compared to the number of studies made on the wicked pipes. The studies on the wickless heat pipes are almost at the early stages.

The number of studies made and the codes developed are also limited. On the other hand, most of the studies made up to now are also mostly inaccurate. This is because the phase change is not taken into consideration. Therefore, for finding the exact thermal performance of the system, the phase change must be considered which is possible with the help of CFD.

2 Existing Method

INTERNATIONAL JOURNAL OF RESEARCH IN MECHANICAL, MECHATRONICS AND AUTOMOBILE ENGINEERING (IJRMMAE)
ISSN: 2454-1435 (Print) | 2454-1443 (online)
Volume 2 Issue 5 – www.ijrmmae.in – Pages 1-8

When the fluid moves because of natural convection and not by any external means, then it is said to be pool boiling. The intention of pool boiling is to convert a portion or the whole of the liquid phase into vapour phase.

In this type, when heat applied to the liquid is below saturation temperature, the liquid boils and moves up. Then, the boiled liquid returns because of natural convection. This is followed by the formation of bubbles that rise and move towards the top of the pool and it is known as sub-cooled boiling. But, as soon as the liquid attains its saturation temperature, the bubble formation will eventually increase and reach the top of the pool. This is saturated boiling.

When the heat pipe is filled with water and heat is supplied to it, the heat transfers from the wall of the heat pipe to the liquid because of convection. Once the saturation temperature is reached, the water moves to the top surface of the pool. Here, the excess temperature will be very low and there will not be any formation of bubbles.

The excess temperature plays a vital role in categorizing pool boiling regimes into different types. The observed different types of pool boiling regimes are natural convection boiling, nucleate boiling, transition boiling and film boiling.

After natural convection boiling, the applied excess temperature will result in the formation of the bubbles. The formed bubbles tend to rise to the top of the pool where it merges together and form slugs of vapour. The formation of vapour bubbles near the heating surface will lower the efficiency of convection process. Nucleate boiling regime serves as the most important regime in pool boiling because a high heat transfer rate can be achieved with small value of excess temperature.

The flow in the condenser section of thermosyphon is laminar and so Nusselt filmwise condensation correlation is used to predict the heat transfer coefficient of the condensation process in the condenser section. The liquid film formed on wall of condenser falls down because of gravity. During this process, latent heat of vapourisation is released by the vapour.

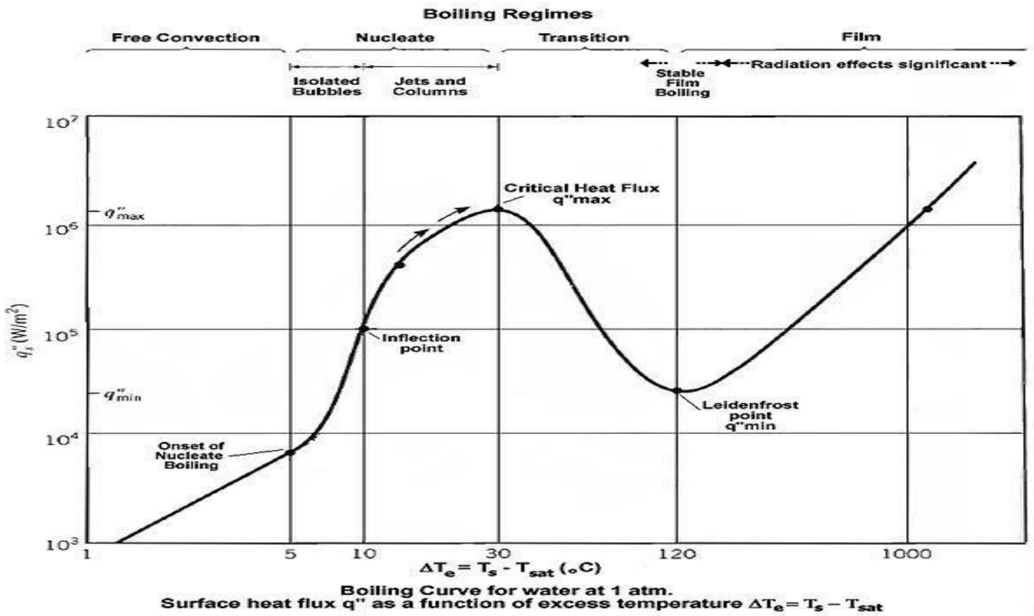

Figure 2: Pool boiling regimes and boiling curve for water

A heat pipe is a two-phase heat transfer device that has an evaporating and condenser section. There is also an adiabatic section provided in between them as shown in the Figure 3.

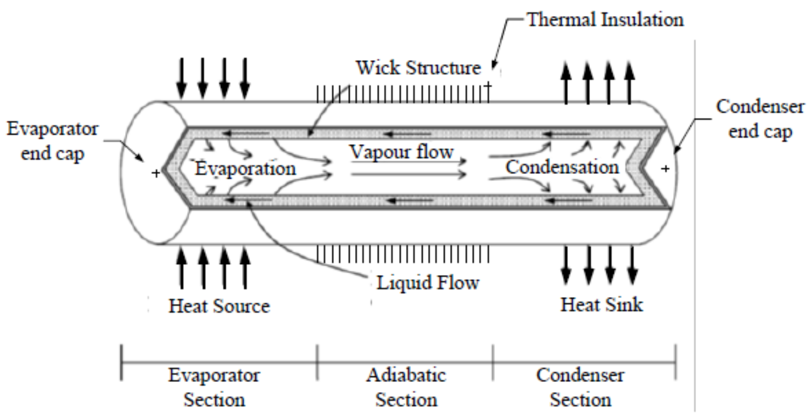

Figure 3:Schematic diagram of a heat pipe showing its components

The heat pipe will be filled with the working fluid. In most of the cases, water is used as the working fluid and it will be at saturated condition. As the system is a closed system, only pure liquid and vapour phase can exist within the heat pipe. When heat is applied, the liquid pool in the evaporator section boils and turns into vapour. The pressure of the resulting vapour makes it move to the condenser section through the adiabatic section. Because of the lower temperature existing at the condenser section, the vapour loses its latent heat at the condenser walls and the condensed liquid is sent back to the evaporator section either by using the capillary wicks or by means of gravity.

This cycle will continue further. The main advantage of heat pipe is that the temperature difference needed to transfer heat is very less when compared to the other systems. Thus, heat pipes have a high effective heat transfer rate.However, the size, material, construction and the working fluid of the heat pipe plays a vital role in the performance of heat exchangers.

3 Proposed System

Numerical modelling uses a set of mathematical correlations for describing the dynamic behaviour of a real system. Numerical models have been extensively used in engineering applications for so many purposes like interpretative, design and predictive studies. Interpretation is applied when numerical models are necessary to explain the logic behind field or laboratory data. Design is applied when numerical models are used to compare the relative performance of different systems without emphasising much on the final predicted performance. Prediction is applied when numerical models are used to reflect the efficacy of the experimental data of the actual system.

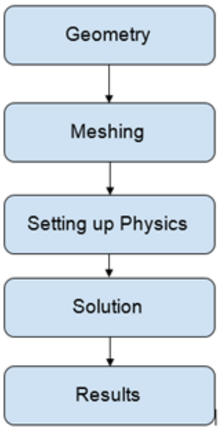

Figure 4: Process of CFD

Validation processes involve an iterative procedure in which the model outcomes are compared with experimental data, until both the numerical modelling and experimental measurements give good accuracy.

CFD serves a greater tool in solving a set of mathematical equations for predicting the approximate solutions of fluid dynamics, heat and mass transfer, thermodynamics, chemical reactions and other related phenomena. The solvation of heat and mass transfer and fluid flow problems can be broadly classified into three types. They are analytical approach, experimental approach and computational approach.

In the first method, that is, analytical approach, continuity, momentum and energy governing equations are used for finding the analytical solutions. But, these analytical solutions can only applicable for a limited range of problems and the derivation efforts require a lot of substantial mathematical efforts. Although they have such a limitation, analytical approaches serve as a vital tool to help engineers understand the fundamental laws that control the behaviour of many applications. In addition to this, the analytical approach can be used as the first step for validating CFD models as well.

In the next type, experimental approach, the physical phenomena of particular applications are predicted with the help of related equipment and measurement tools. But the whole experiments take a long time for setting-up and to run, and they are technically difficult as well or will be more expensive to perform. The results of experimental approaches provide more reliable solutions as computational models require reliable experimental data for validation.

In the third approach, computational approach, CFD techniques have been developed and serve a major purpose in solving complex flow problems for a variable number of engineering applications due to their universality, flexibility, accuracy and efficiency. They are also capable of providing a complete set of relevant information through a particular domain, which is not possible in the experimental approach.

CFD works on the basis of finite volume method that solves numerically the discretised form of the governing equations for acquiring solution fields describing the fluid dynamics throughout the computational domain.

4 Results and Discussion

Heat transfer process

For the observing of the heat transfer process during the thermosyphon operation, the temperature contours at different times have been observed during the start-up (heating) and steady-state operation. In this visual observation, the temperature distribution in the fluid region inside the evaporator, adiabatic and condenser sections have been recorded for different fill ratios. Two fill ratios (0.5 and 1.0) have been selected to demonstrate the heat transfer process for water. A heating power of 173 W is selected to compare the heat transfer process for both fill ratios.

At the beginning, the temperature in the evaporator section increased due to the constant heat flux throughout, which allows heat to transfer through the evaporator wall into the water pool. Due to the temperature difference between the wall and the working fluid within the thermosyphon, boiling heat transfer continues on the walls of the evaporator section. The region of high temperature in the evaporator section expands due to the vapour moving upward. As the heating power in the evaporator section continues, the vapour flows across the adiabatic section to the condenser section. Then, a high temperature region appears in the condenser section due to the vapour reaching this section. As the vapour condenses along the inner surface of the condenser wall, a lower temperature is seen near the inner surface of the condenser wall.

With the help of gravity, the condensed liquid falls back to the evaporator section. The above cycle describes the process of heat transfer during the operation of the thermosyphon. After that, the temperature distribution inside the thermosyphon becomes uniform. A similar heat transfer process for the water filling the total volume of the evaporator section (fill ratio 1.0) is observed to those of the half-filled evaporator section.

Evaporation process

The pool boiling phenomenon taking place inside the evaporator section has also been visualisedfor a heating power of 173 W. A red colour illustrates the presence of only vapour (vapour volume fraction

INTERNATIONAL JOURNAL OF RESEARCH IN MECHANICAL, MECHATRONICS AND AUTOMOBILE ENGINEERING (IJRMMAE)
ISSN: 2454-1435 (Print) | 2454-1443 (online)
Volume 2 Issue 5 – www.ijrmmae.in – Pages 1-8

= 1) while a blue colour stands for the presence of only liquid (vapour volume fraction = 0). At the beginning of the process, the liquid pool that initially filled half of the evaporator for FR = 0.5, and filled the total volume of the evaporator for FR = 1.0 was heated by imposing a constant heat flux into the wall of the evaporator section. Heat is then conducted through the evaporator wall to the inner wall to be transferred into the saturated liquid by boiling.

At positions where the liquid temperature exceeds the saturation temperature, the liquid begins to boil and phase change occurs. Hence, nucleation sites take place and vapour bubbles start to form at those. Isolated vapour bubbles are formed due to continuous nucleation and transported toward the top region of the water pool where they break up and release their vapour content, as can be seen in the figures. This continuous boiling of liquid results in a decrease in the liquid volume fraction and an increase of the vapour volume fraction.

Condensation process

Following the evaporation process, where the liquid pool of the working fluid changesto vapour, the converse process takes place in the condenser section. Figure 3.12illustrates the liquid film condensation phenomena that occur along the inner surface ofthe condenser wall and focus is made on the condensed liquid film region in the lowerpart of the condenser. After the boiling process, saturated vapour is transported upwardto the condenser. As the vapour reaches the condenser's wall, where a convection heattransfer boundary condition is defined, the vapour condenses along the cold wallsforming filmwise condensation as shown in Figure 3.12. As the condenser section isplaced above the evaporator, the liquid film is returned by gravity and recharges thewater pool in the evaporator section.

INTERNATIONAL JOURNAL OF RESEARCH IN MECHANICAL, MECHATRONICS AND AUTOMOBILE ENGINEERING (IJRMMAE)
ISSN: 2454-1435 (Print) | 2454-1443 (online)
Volume 2 Issue 5 – www.ijrmmae.in – Pages 1-8

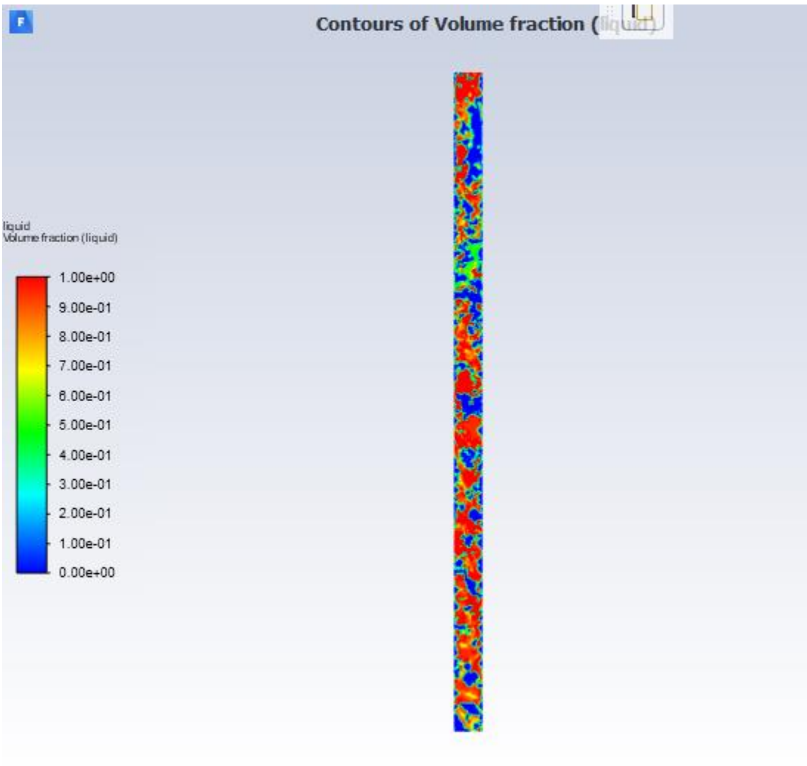

Figure 5: Evaporation and Condensation Process in thermosyphon

5 Conclusion & Future Work

While comparing fill ratios of 0.5 and 1, the high efficiency is achieved in 0.5 fill ratio. This is because the fill ratio of 1 attains flooding limit as the evaporator section is fully filled with water phase. Whereas in fill ratio of 50 %, there is enough time to boil and condensate. So, the process here is cyclic and it works in a proper limit which gives us the conclusion that 0.5 fill ratio is more efficient than FR 1.

Heat pipe technology is receiving increased attention nowadays and the optimisation of heat pipe performance is of great interest. Wickless heat pipe systems are more desirable than conventional systems due to their passive operation, reliability, efficiency and the cost and ease of manufacturing. However, up to now, computational studies developed on wickless heat pipes, displaying the complex two-phase flow inside the heat pipe are at an early stage. In this study, a novel CFD model has been developed that allows the detailed study of the two-phase flow and heat transfer phenomena during the operation of a wickless heat pipe. Working fluid water was investigated. This project describes the CFD procedure employed to study the evaporation and condensation phenomena and evaluate the performance of these fluids as working fluids in the wickless heat pipe. Temperature profiles along the wickless heat pipe have been determined and the two-phase flow characteristics have been visualised.

The ANSYS FLUENT CFD simulation package was used to visualise the evaporation, condensation and heat transfer processes during the start-up and steady state process of a wickless heat pipe. The phase change process occurring during the evaporation and condensation was simulated in FLUENT, due to the flexibility and accuracy of the finite volume method.

6 References

[1]. CFD modeling of flow and heat transfer in a thermosyphon- Asghar Alizadehdakhel, Masoud Rahimi, Ammar Abdulaziz Alsairafi.
[2]. Modelling of the Thermal behaviour of Two-phase Thermosyphon - Bandar Fadhl.
[3].Computational Fluid Dynamics Analysis of Two-phase Thermosypho - Nikhil E Choudhari, Nishtha Vijra, T.P.Singh.
[4].CFD Analysis and Visualization of the TwoPhase Flow in a Thermosyphon for a Passive Heat Removal System of a Nuclear Power Plant - Joseph Seo, Jae Young Lee.
[5].Heat transfer capability of Thermosyphon heat transport device with experimental and CFD studies - B.M. Lingadea, Elizabeth Rajub, A Borgohain , N.K. Maheshwari , P.K.Vijayan.
[6].Experimental and computational analysis of thermosyphon based water heaters - Siddarth Ashokkumar
[7].R.S. Gauglar, Heat transfer device, US patent 2350348, 1942.
[8].G.M. Grover, Evaporation–condensation heat transfer device, US patent 3229759, 1963.
[9].G.P. Peterson, An Introduction to Heat Pipes, Modeling, Testing and Applications, John Wiley & Sons Inc., New York, USA, 1994.
[10]. A.A. Mohamad, I. Sezai, Natural convection in C-shaped thermosyphon, Numerical Heat Transfer. Part A, Applications 32 (3) (1997) 311–323.