COPY RIGHT

Paper Authors **M.Sandeep, A.Abinay, CH.Pavan, CH.Ajay, M.Srikanth**

.

USE THIS BARCODE TO ACCESS YOUR ONLINE PAPER

To Secure Your Paper As Per UGC Guidelines We Are Providing A Electronic Bar Code

THERMAL ANALYSIS OF CLOSED LOOP HELICAL PULSATING HEAT PIPE

M.Sandeep[1*], A.Abinay[2], CH.Pavan[3], CH.Ajay[4], M.Srikanth[5]

ACE Engineering College, Ankushapur, Ghatkesar, Telangana-501301[12345], Department of Mechanical Engineering, ACE Engineering College, Ankushapur, Ghatkesar, Telangana 501301

ABSTRACT

The computational investigation presented in this study focuses on understanding the heat transfer properties of a closed-loop pulsating heat pipe (CLPHP). In contrast to conventional heat pipes, the CLPHP examined in this research does not incorporate a wick material. This simplification facilitates easier modeling and analysis of the CLPHP's behavior. To assess the CLPHP's performance, flow visualization techniques were employed using ANSYS Fluent R22 software. Realistic operational conditions were simulated by applying appropriate boundary conditions to the CLPHP. The study employed water-water vapor and ethanol as the working fluids.

In summary, this computational study offers valuable insights into the heat transfer characteristics of a closed-loop pulsating heat pipe (CLPHP). By visualizing the flow patterns, observing phenomena like nucleation boiling and slug formation, and analyzing the response to varying heat flux inputs, researchers can enhance their understanding of the CLPHP's performance and its potential for efficient heat transfer in diverse applications.

INTRODUCTION FOR PULSATING HEAT PIPE:

The continuous advancement of electronic components, aiming for higher speed and smaller size, has resulted in a growing challenge in terms of thermal management. Proper thermal management is crucial to ensure that the operating temperatures of these components remain within a safe range. To assess the performance of closed-loop pulsating heat pipes (CLPHPs), computational fluid dynamics (CFD) is employed. The geometry of the CLPHP is modeled using a fluid flow analysis system, such as ANSYS Fluent, within the design modeler. After creating geometric models and generating a mesh, the model is analyzed by varying the wall heat flux at the evaporator for a specific filling ratio. Two different cases of the working fluid are considered: water-water vapor and ethanol.

In summary, computational fluid dynamics is utilized to evaluate the performance of closed-loop pulsating heat pipes. The modeling process involves creating the geometric design and generating a mesh, followed by analyzing the model while considering variations in wall heat flux at the evaporator for a specific filling ratio. The study considers both water-water vapor and ethanol as the working fluids.

CFD PROCEDURE:

For numerical analysis in CFD, it requires five stages such as:

- Geometry creation
- Grid generation
- Flow specification
- Calculation and numerical solution
- Results.

DESIGN:

To begin, open ANSYS Workbench and access the "ANSYS Fluent Launcher" interface. From there, select the "Fluid Flow (Fluent)" option, which provides access to the features and functions of ANSYS Fluent for simulating fluid flow and analyzing results. Next, right-click on the geometry tab and choose "New Design Modeler Geometry". Select the ZX plane and set the desired units for the geometry.

To sketch the geometry, use the circle tool and draw a circle with a dimension of 2mm. Set the distance between the Z-axis and the center of the circle to 3mm. Then, in Sketch 2, draw a line measuring 9.5mm. Right-click and choose "Generate". Proceed to the "Sweep" operation, selecting the circle as the profile and the line as the path. Adjust the operating material to "Add Frozen" and specify the number of turns as 3.5mm. Generate the geometry accordingly.Moving on to Plane 4, modify the transform option to "Offset Z" with a value of 30mm. Generate the geometry based on these settings. Create a body transformation by clicking "Create" and selecting "Body Transformation". Apply the mirror option and choose Plane 4 as the mirror plane. Generate the transformation.

In summary, the steps involve opening ANSYS Workbench, accessing the ANSYS Fluent Launcher, creating and modifying geometry using various tools and operations, and generating the geometry based on the specified settings.
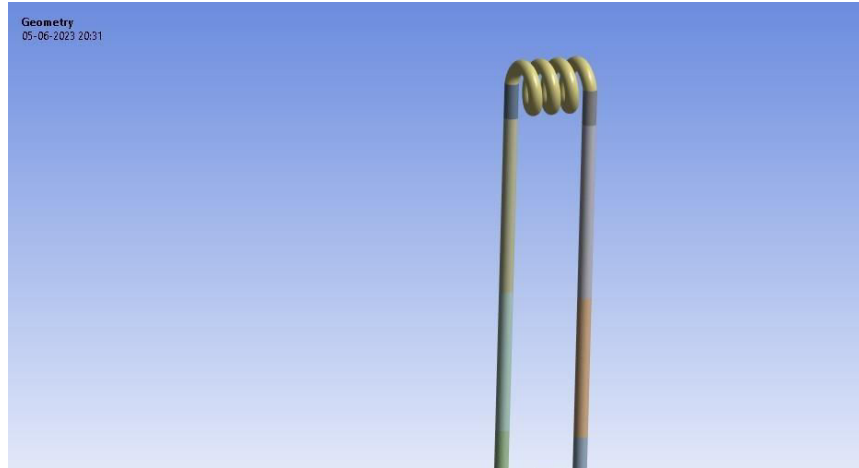

Fig 1: Schematic view of closed loop pulsating heat pipe

MESHING:

In addition to the automated settings, ANSYS Meshing provides additional control with the option to specify combinations of point controls, edge controls, surface controls and/or body controls. In this case, the automatic method for mesh shape is selected; however the sizing for the mesh is done manually. The mesh sizes are generated automatically as shown in fig. With this control setting, meshing is generated with 30,450 number of nodes for air and 22,736 numbers of nodes for water and 26,325 number of elements for air and 18,630 number of elements for water.
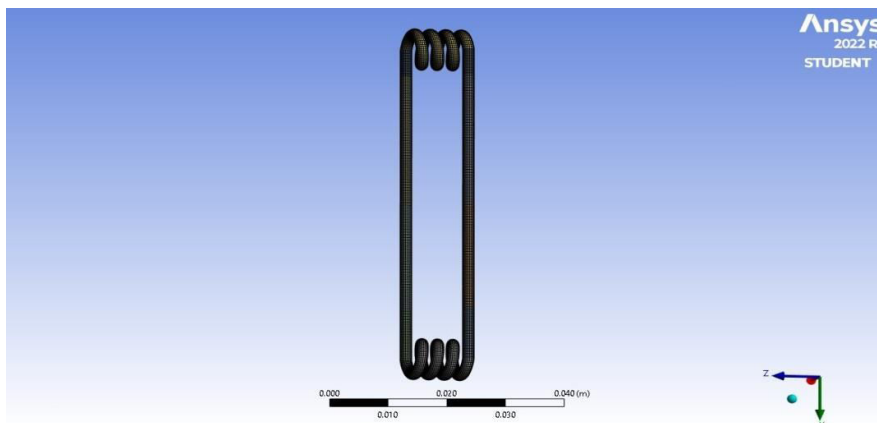
Fig 2: Meshing of pulsating heat pipe

GENERAL PROPERTIES:

When open the setup of the CFD click on general, then click on check to Check the mesh so that we can observed mesh checking in console sheet. Then click on pressure-based type solver and then click on absolute velocity formulation. The transient model was chosen since there is a dependence on time and there is no steady state for a pulsating heat pipe. The gravity was set as 9.81 m/s² in the negative y direction. Nothing else was changed from the default

SETTING UP THE MODELS:

The flow inside a pulsating 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 thephases, which is very important to describe this flow.

SETTING UP THE VISCOUS MODEL:

➢ Use the default Standard from the k-epsilon Model list.
➢ Select Enhanced Wall Treatment for the Near-Wall Treatment.

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 400kpa The other parameters used are the default ones.

BOUNDARY CONDITIONS:

The boundary conditions are introduced to ANSYS. Named selections shouldbe created during the meshing to facilitate this step. After clicking on "Boundary Conditions". The boundary conditions are defined. As an example, let's set the evaporator conditions. Clicking on "evaporator", its type is "wall", then clicking on edit,The Wall Thickness used here is 0.5 mm

RUNNING THE SIMULATION:

The biggest issue with simulation of multiphase flows is that the time step needs to be sufficiently small to capture the movement of the particles, and at the same time it needs to be big to reduce the computational time.

RESULTS AND DISCUSSION:

Post-processing in Ansys software refers to the analysis and visualization of computational fluid dynamics (CFD) simulation results after the completion of a simulation run. Ansys provides several tools and functionalities for postprocessing CFD data, allowing engineers and researchers to gain insights into the flow behavior, analyze variables of interest, and make informed decisions based on the simulation results.

POST PROCESSING OF COPPER MATERIAL WITH WATER:

The diagrams indicate about pressure, temperature and velocity of the fluid flow in the heat pipe by line and plane cut methods. The "charts" typically refers to graphical representations or plots of various data or results obtained from post-processing simulations.
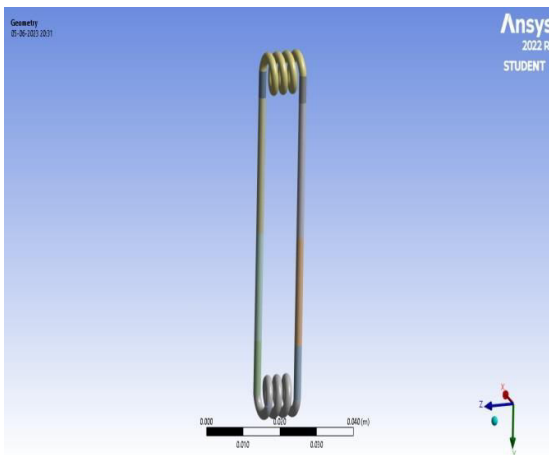
Fig 3: Co ordinate of CLPHP
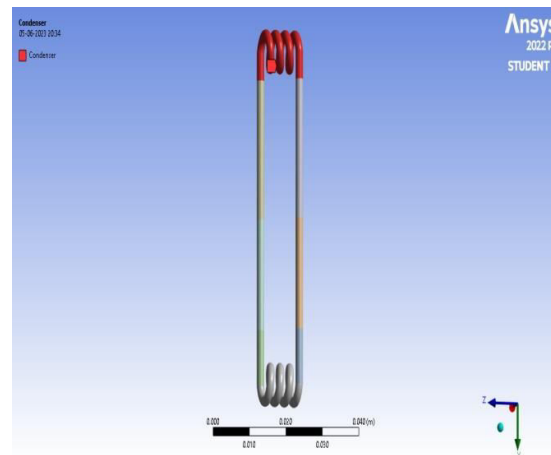
Fig 4: Geometry with condenser
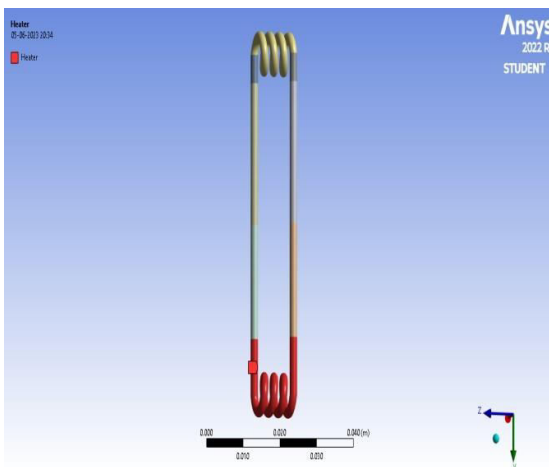
Fig 5: Geomerty with heater
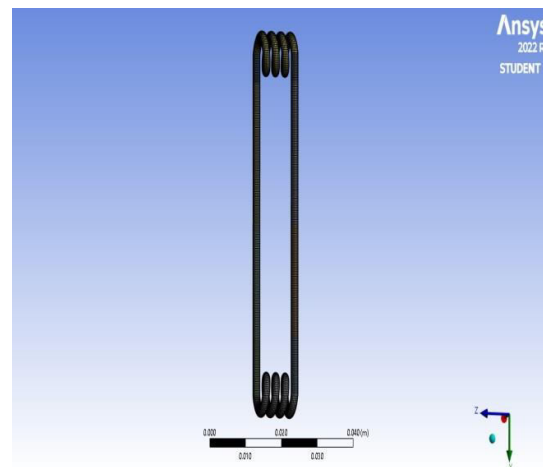
Fig 6: Geomerty with Mesh
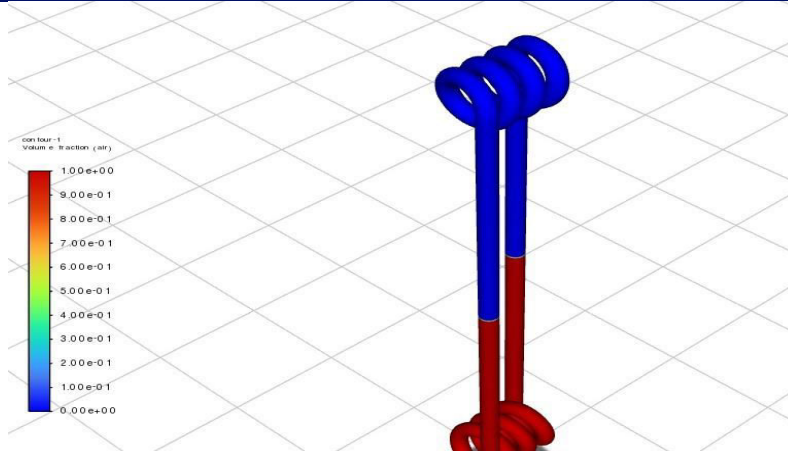
Fig 7: Contour volume fraction of air with copper material
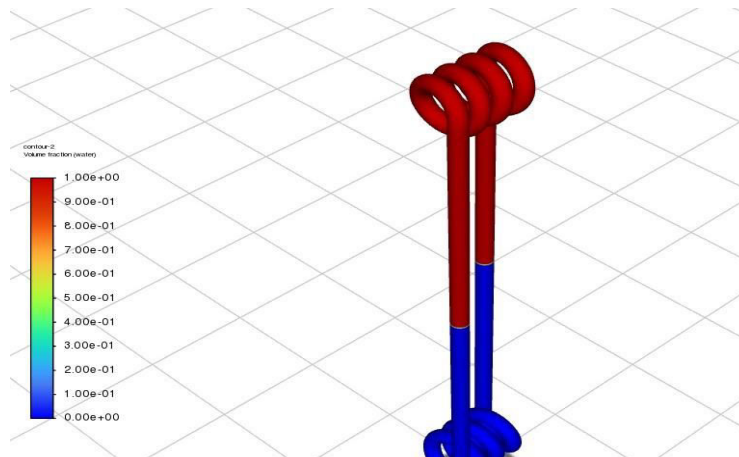
Fig 8: Contour volume fraction of water with copper material
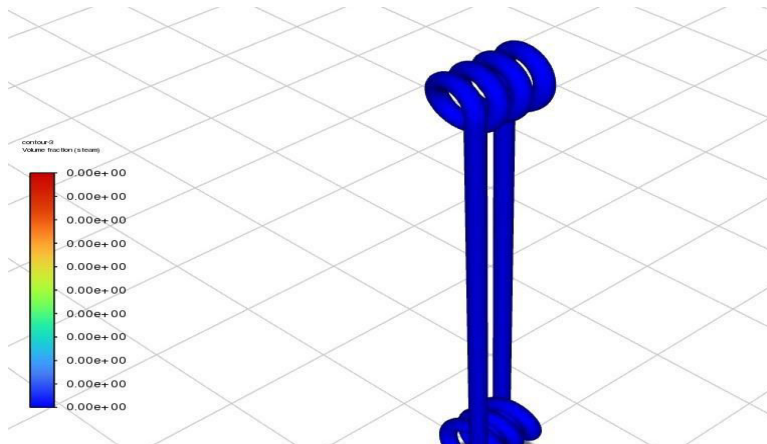
Fig 9: Contour volume fraction of steam with copper
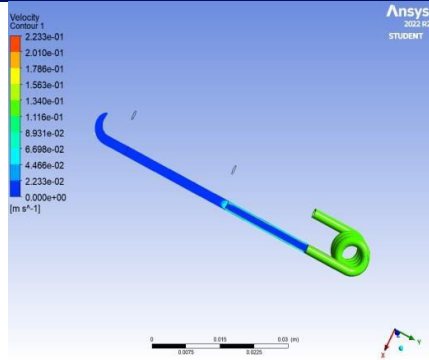
Fig 10: Velocity contour
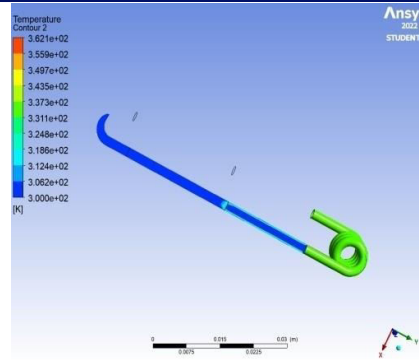
Fig 11: Temperature contour
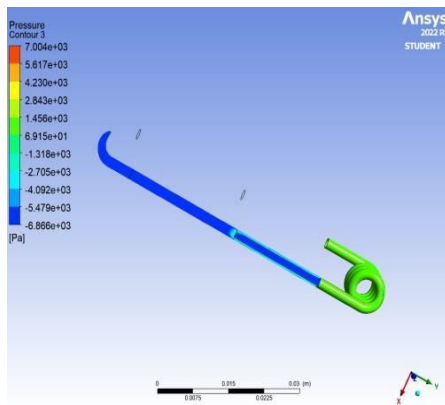
Fig 12: pressure contour
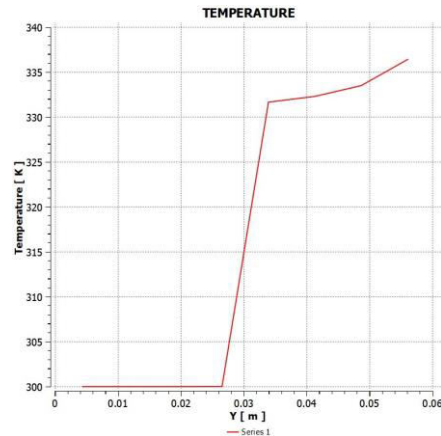
Chart1: Temperature chart
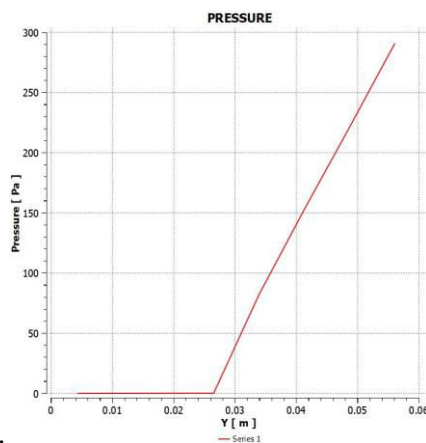
Chart2: Pressure chart
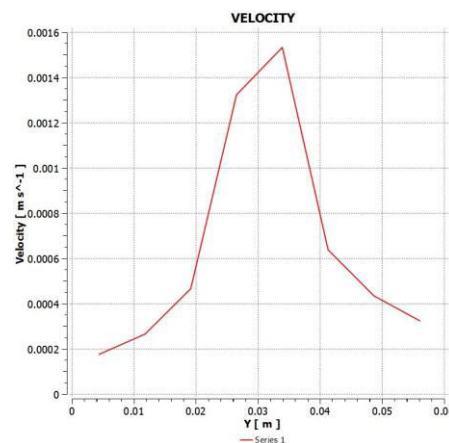
Chart3: Velocity chart

NICKEL WATER POST PROCESSING:
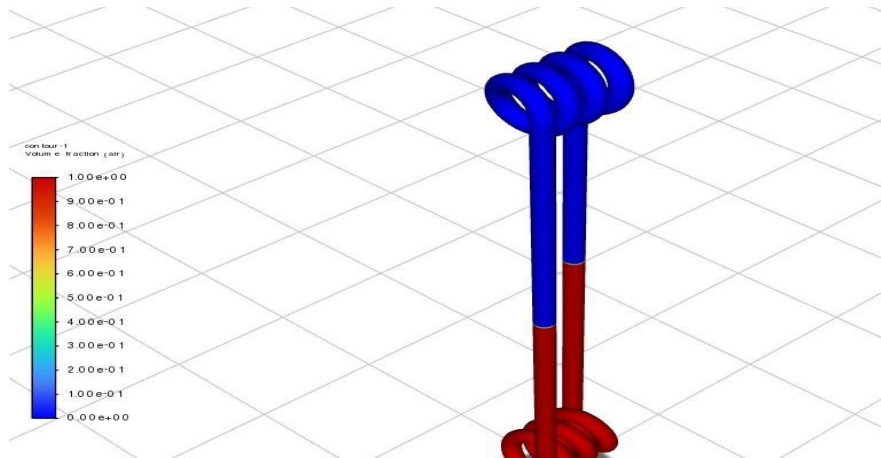
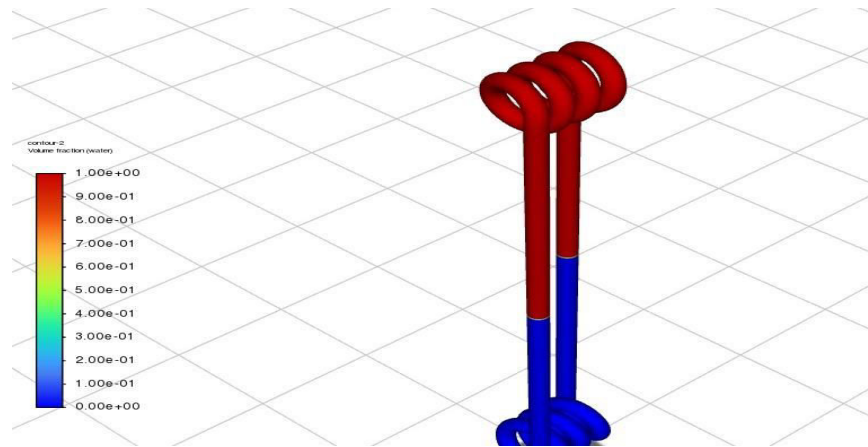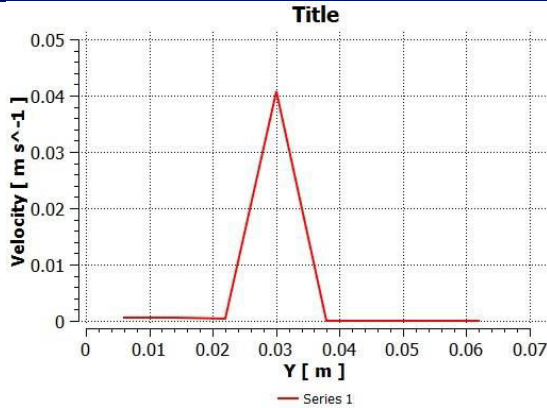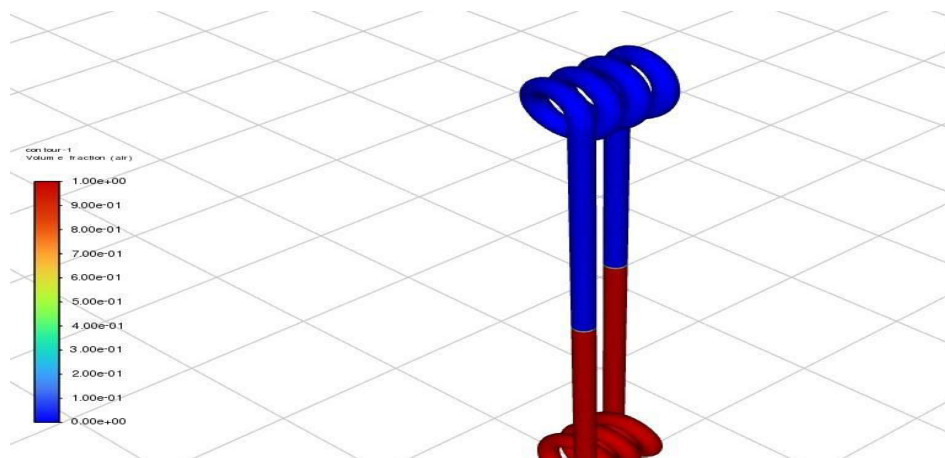Fig 13: Contour volume fraction of air with nickel material

Fig 14: Contour volume fraction of water with nickel material
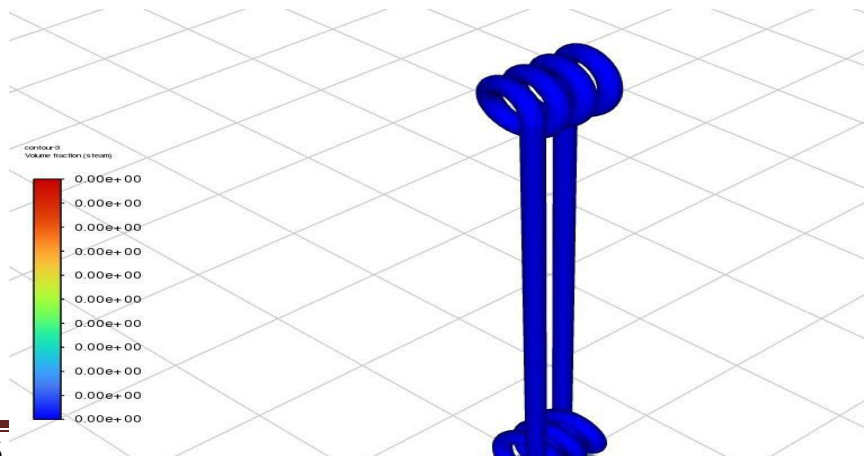
Fig 15: Contour volume fraction of steam with nickel

Chart 4: Velocity chart
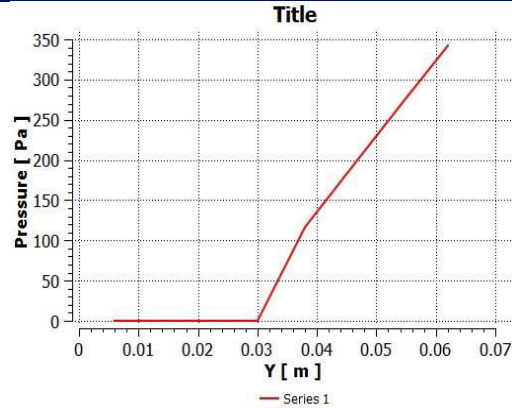
Chart 5: Pressure chart
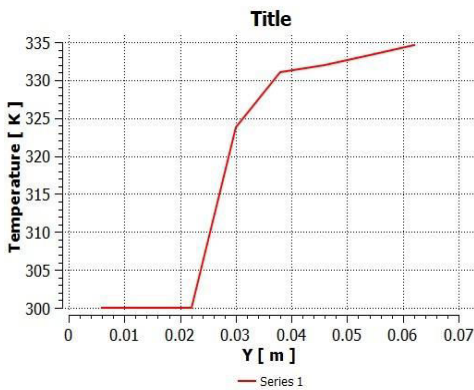
Chart 6: Temperature chart
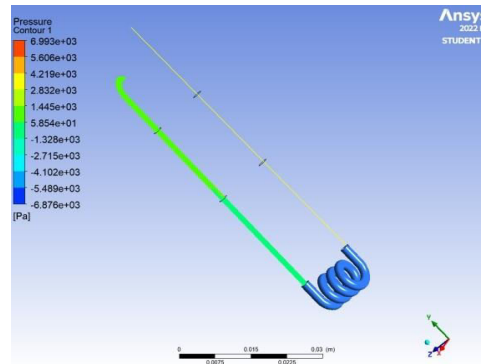
Fig 16: Pressure contour
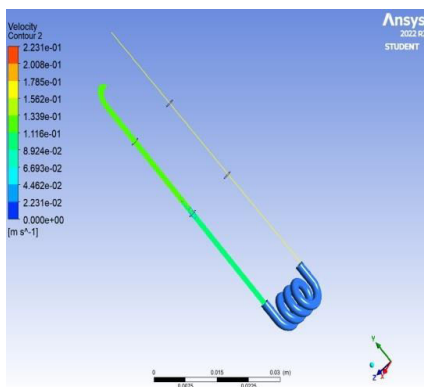
Fig 17: Velocity contour
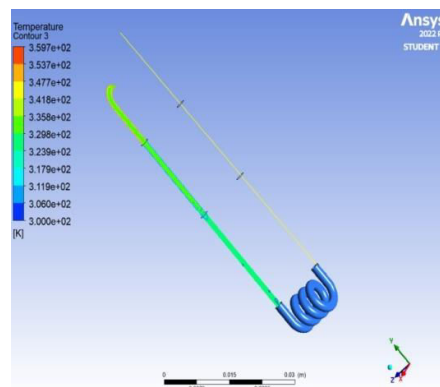
Fig 18: Temperature chart

NICKEL ETHANOL POST PROCESSING:

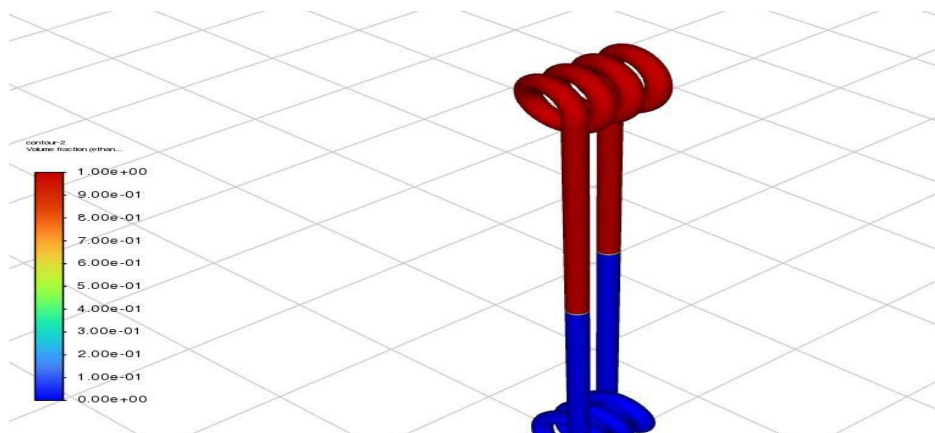Fig 19: Contour volume fraction of air with nickel material(ethanol)

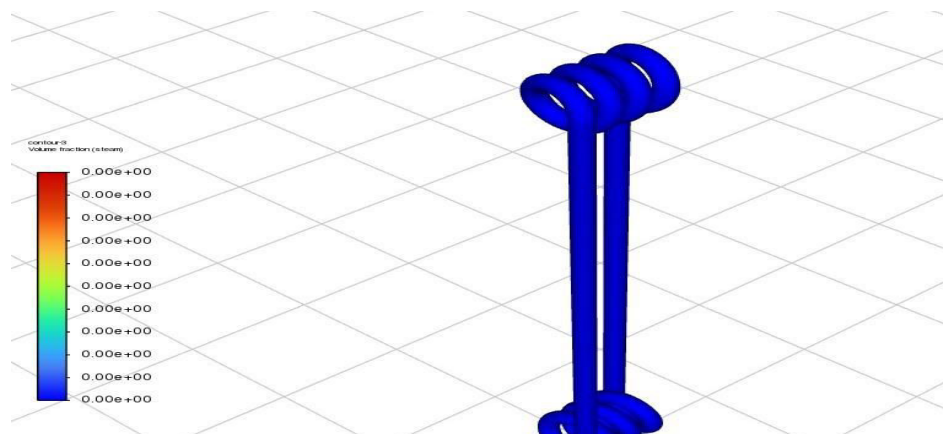Fig 20: Contour volume fraction of water with nickel material(ethanol)

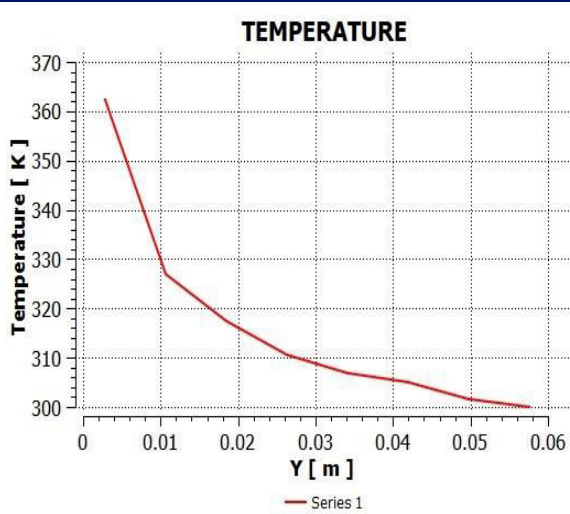Fig 21: Contour volume fraction of steam with nickel material(ethanol)
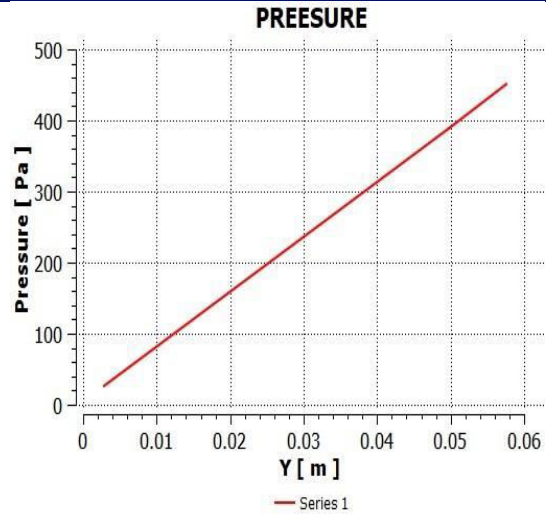
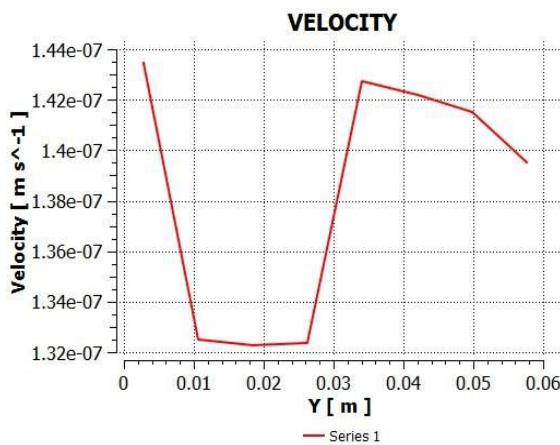Chart 7: Temperature chart
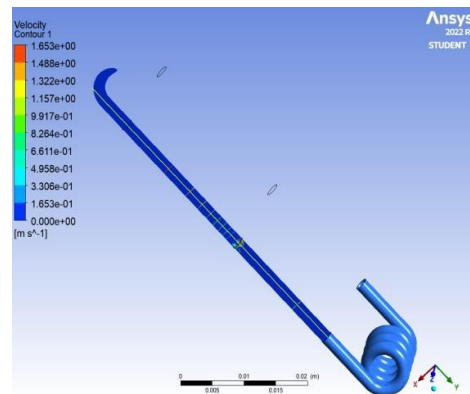

Chart 8: Pressure chart
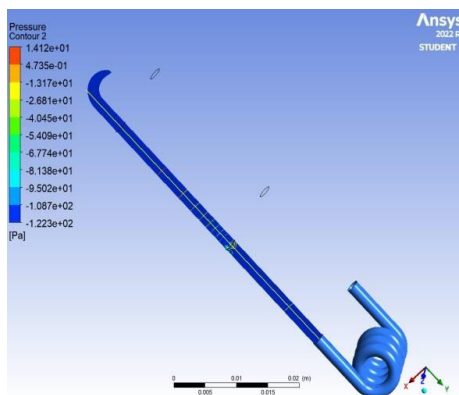

Chart 9: Velocity chart
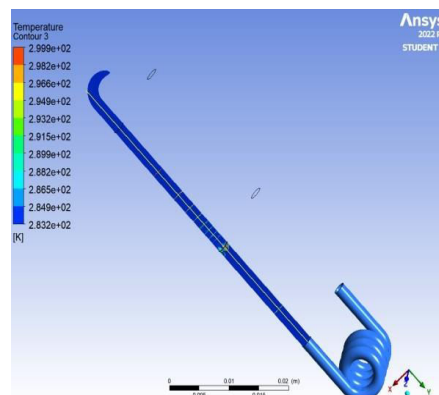

Fig 22: velocity contour

Fig 23: Pressure contour

Fig 24: Temperature contour

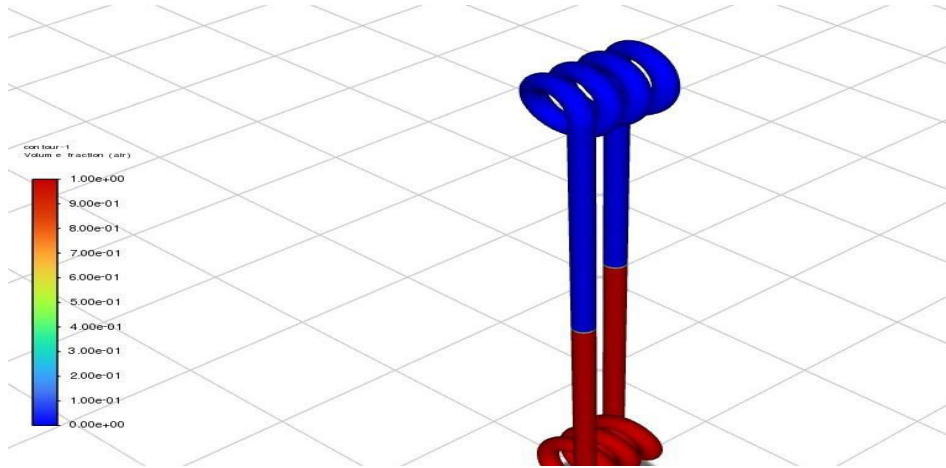**COPPER MATERIAL WITH ETHANOL:**

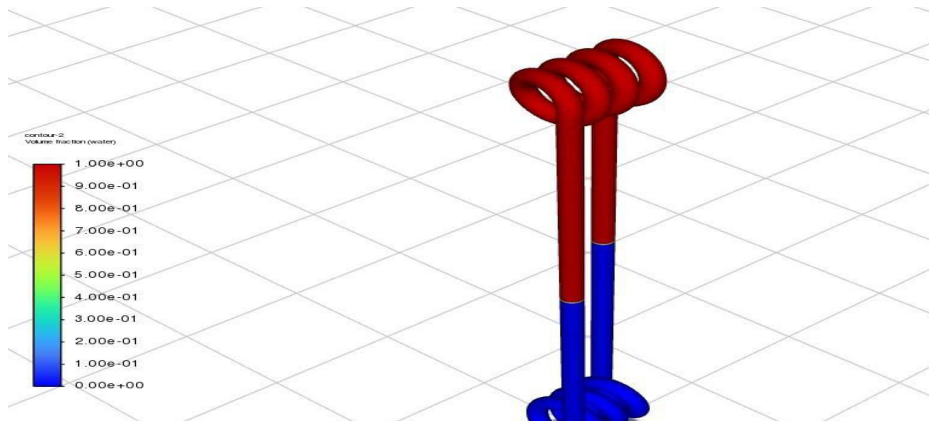Fig 25: Contour volume fraction of air with copper material(ethanol)

Fig 26: Contour volume fraction of water with copper material(ethanol)
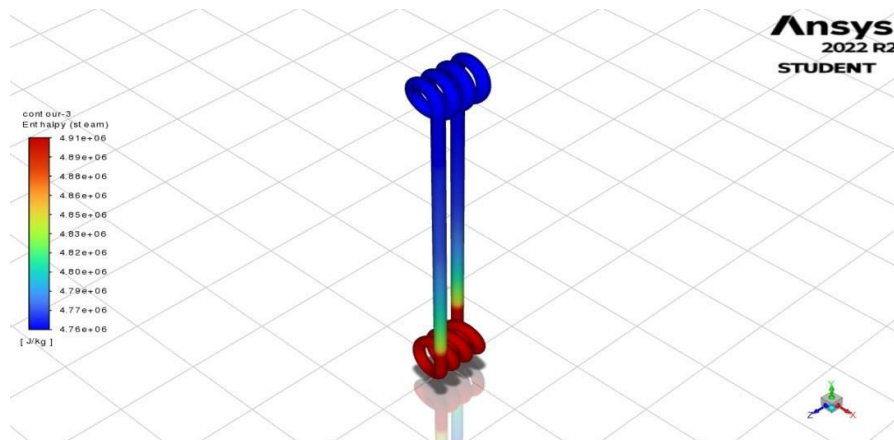

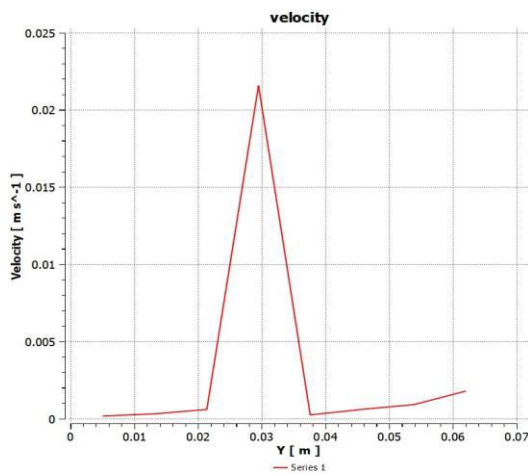Fig 27: Contour volume fraction of steam with copper material(ethanol)
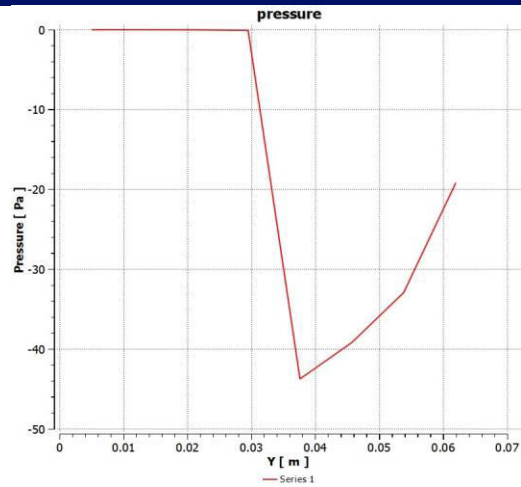
Chart 10: Velocity chart
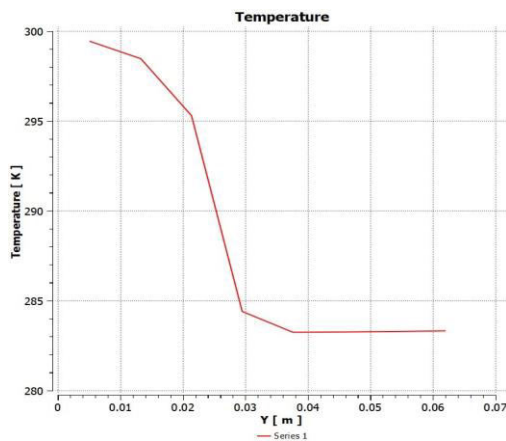

Chart 11: Pressure chart
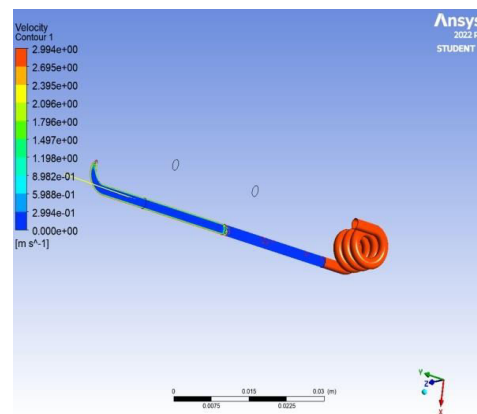

Chart 12: Temperature chart
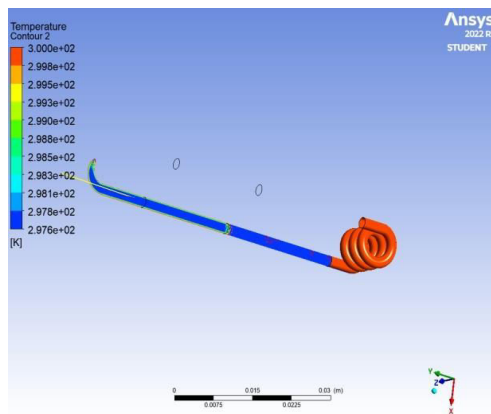

Fig 28: velocity contour

Fig 29: Temperature contour
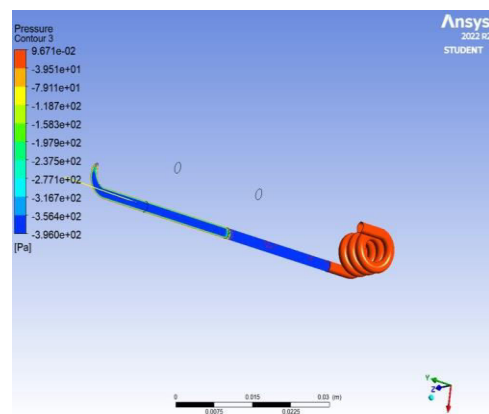
Fig 30: Pressure contour

THERMAL TRANSIENT

Thermal transient diagrams, also known as transient temperature or heat flux plots, provide valuable information about the behavior of temperatures or heat fluxes over time during a transient thermal analysis. These diagrams are commonly used in engineering and scientific fields to study the thermal response of a system to changes in boundary conditions or heat sources.

Here's what thermal transient diagrams can indicate:Internal heat generation, Heat flux, Total heat flux, Directional heat flux, Heat flow, and Temperature.

Copper Material Results:

Fig 31: Temperature Fig 32: Total heat flux Fig 33:Directional heat flux

Nickel Material Results:

Fig 34:Temperature Fig 35: Total heat flux Fig 36:Directional heat flux

CONCLUSION

➢ Based on application we have concluded that copper material and ethanol as working fluid is more relevant for the pulsating heat pipe for electronic device.
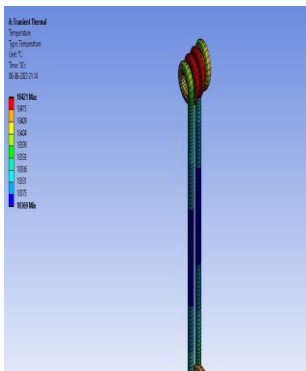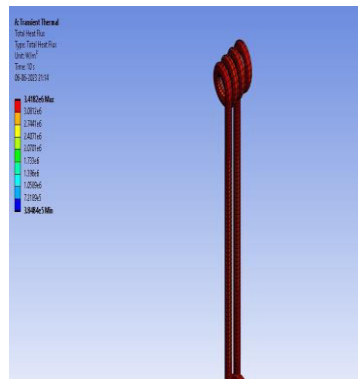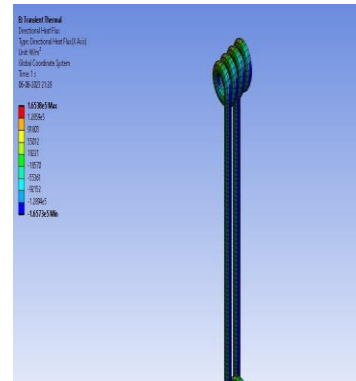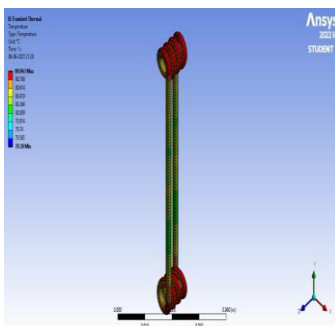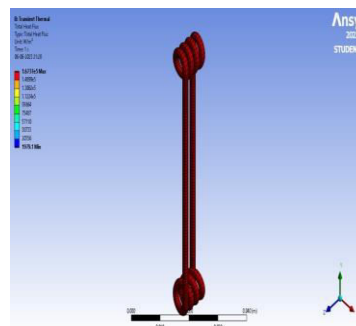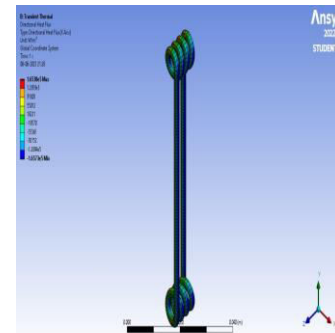
➢ Using the computational fluid dynamics (CFD) approach, this study examines the flow and heat transfer phenomena occurring within a closed-loop pulsating heat pipe. Furthermore, the impacts of the heat flux applied to the evaporator and the utilization of various working fluids are investigated. The simulations conducted in this research yield several noteworthy findings, which are summarized as follows.

➢ Firstly, the heat transfer rate between water and ethanol exhibits minimal disparity. Both fluids demonstrate comparable abilities to facilitate heat transfer within the system.

➢ Secondly, when ethanol is employed as the working fluid, the onset of oscillation occurs more rapidly compared to water. This implies that the pulsating motion of the fluid begins earlier when ethanol is utilized.

➢ In conclusion, the results obtained from this investigation, obtained through the application of CFD methodologies, offer valuable insights into the behavior of closed-loop pulsating heat pipes. The study reveals that the heat transfer rates of water and ethanol are practically indistinguishable. Additionally, it is observed that ethanol exhibits an advantage in terms of the initiation time for oscillation.

FUTURESCOPE

The future scope of thermal analysis of pulsating heat pipes (PHPs) is quite promising. Here are a few potential areas of development and research:

➢ Advanced Materials: Researchers are exploring new materials with enhanced thermal properties to improve the performance of PHPs. This includes using nanofluids, hybrid fluids, and advanced coatings on the inner walls of the pipes. Future studies may focus on developing materials with higher thermal conductivity, reduced evaporation and condensation losses, and improved wettability.

➢ Miniaturization and Microscale PHPs: PHPs have predominantly been studied and applied in macro-scale systems. However, there is growing interest in miniaturizing PHPs for applications in microelectronics cooling and other small-scale devices. Future research may focus on understanding the unique thermal phenomena and challenges associated with microscale PHPs, such as capillary forces, surface tension effects, and interfacial interactions.

REFERENCES

[1] H. Akachi, "U.S. patent 4921041," 1990.

[2] S. Khandekar, S. Manyam, M. Groll, and M. Pandey, "Two-phase Flow Modeling in Closed Loop Pulsating Heat Pipes," in Proc. 13th Int. Heat Pipe Conf., Shanghai, China, 2004.

[3] "Introduction to Multiphase Flow Modeling in ANSYS FLUENT," LearnCAx.

[4] A. K. Singh, "Numerical Analysis of Performance of Closed Loop Pulsating Heat Pipe," 2013.

[5] C. Fasula, "OSCILLATING HEAT PIPES (OHP)," 2009.

[6] P. Charoensawan, "Heat Transfer Characteristics of Closed Loop Oscillating Heat Pipes," Ph.D. Thesis, Chiang Mai University, Chiang Mai, Thailand, 2003.

[7] Y. Yanfenghai, "Experiment of Thermal Performance of Oscillating Heat Pipe under Various Conditions," 2020.

[8] B. Sangar, "Investigation on Heat Pipes using Different Concentrations of Nanofluid," 2020.