ISSN 2229-5518

CFD Analysis of Single turn Pulsating Heat Pipe

J.Venkata suresh, P.Bhramara

Abstract— Pulsating Heat Pipe (PHP) is a heat exchanger device which absorbs heat from evaporator region and transfers it to the condenser region. The flow in pipe is Multi-Phase flow. Vapour plugs and Liquid slugs are formed in PHP due to the capillary action.CFD modeling is done in ANSYS CFX with single turn of PHP and Acetone is used as the working fluid and a fill ratio of 60%. At evaporator boundary, heat flux that is equivalent to 9 W to 15 W is supplied and the condenser boundary is set as heat flux of range 7945 W/m^2 and in the adiabatic section heat flux is zero. The obtained CFD results are compared with the experimental paper [1].The CFD analysis is performed and the outputs of the simulations are plotted in graphs and contours. Decrease in the acetone temperature at the evaporator suggests that heat is carried away to the condenser part. Change in volume fractions of acetone and air in three regions viz. evaporator, adiabatic region and condenser reflects to the flow pattern of the fluid inside the PHP.

Index Terms— Two Phase Flow, Pulsating Heat pipes (PHP), Fill ratio, Orientation, working Fluids, Computational Fluid Dynamics (CFD)

1 INTRODUCTION

Heat pipe was developed in early 1960s, and since then this technology has evolved into many different shapes and forms. Moreover, it has been used in numerous applications from computer cooling to spacecraft thermal control, and includes cooking, cooling of fuel cells, space heating or cooling. Different applications have different structure, size and configuration of heat pipe. Pulsating heat pipe (PHP) is a special type of heat pipe where the driving force is the slug/plug motion of the working fluid in the tube, generated by the evaporation. PHPs consist of a tube bent to form several parallel channels. It can be configured as an open and closed loop. In the first one, one end of the PHP is pinched-off and welded; while the other end presents a service valve for vacuum and charge (this valve can be later removed). The closed loop PHP is an endless tube as both ends are welded together. Each PHP configuration presents particular operation modes, which are mainly guided by the chaotic slug/plug motion. Either PHP configuration presents a high dependence on their thermal behavior related to the gravity vector during operation, which must be carefully considered. Higher operation temperatures are achieved when the PHP operates at the vertical orientation, while at horizontal orientation, the operation temperatures are lower.

Invented in 1990s, pulsating heat pipe (PHP) is a passive heat transfer device that has potential applications in solar cell, fuel cell, space and electronic cooling. PHP, also known as oscillating heat pipe, transfers the heat from its evaporator region to the condenser region. In condenser the heat is dissipated to the sink, which it does by the phase change phenomenon of the working fluid being filled in with certain volume ratio. This device has an ability to transfer a large amount of heat over its length with a small temperature drop.

- Asst. prof. & Reasearch Scholar JNTUCEH, Department of Mechanical Engineering, GRIET, Hyderabad, Telangana ,India, E-mail: venkatasureshj@gmail.com
- Associate professor, Department of Mechanical Engineering, JNTU College of Engineering, Hyderabad, Telangana ,India E-mail: bhramara74@yahoo.com

Pramod R Pachghare et. al. [5] used methanol, ethanol, acetone water and different binary mixtures are used as working fluids with copper tube as PHP material, the filling ratio was 50%,ten turns with the different heat inputs of (10 - 100)W was supplied to PHP were observed, results show that thermal resistance decreases rapidly with increasing heat input supplied from 20-60W. Out of the different working fluids used acetone gives the best thermal performance. Rudranaik, Linford Pinto et. al. [9] explained about steady state experiments for various heat loads and different working fluids like Acetone, Methanol, Ethanol, Heptane and Distilled water. The results showed that Heptane and Acetone exhibit better heat transfer characteristics of PHP.Bandar Fadhl et. al. [7] CFD modeling is used to simulate the two-phase heat transfer mechanisms in a wickless heat pipe with R134a and R404 as working fluids. CFD results were compared with experimental results thus CFD model was successful in reproducing the heat and mass transfer with R134a and R404a charged thermosiphon where nucleate pool boiling in the evaporator section and the liquid film condenser section.

R. Naik et. al. [8] conducted the transient and steady state experiments on a single turn closed PHP. Copper is used as the capillary tube material. Experiments are conducted both in the horizontal as well as in vertical orientations with heat input is given with varying load conditions of 9-15W. Acetone, Methanol and Ethanol are used as the working fluids with different fill ratios from 60% to 80%.from the reults Acetone is the better working fluid among the working fluids in terms of lower thermal resistance and higher heat transfer coefficient, better at fill ratio of 60% respectively. Dharmal A Baitule et. al. [1] conducted the transient and steady state experiments on a two turn closed loop PHP .Copper capillary tube. The experiments are conducted on vertical orientations for different heat input varying from 10W to 100w.Ethanol Methanol, Acetone and water are used as working fluids for different fill ratios from 0 to 100%.The performance parameters like temperature difference between evaporator and condenser thermal resistance and overall heat transfer coefficient are evaluated. Thus the experimental results shows that the Heat transfer characteristics lower thermal resistance and higher heat transfer coefficient of PHP are found to be better at fill ratio 60% for different heat input respectively. Jason Clement et. al. [4] conducted an experimental investigation PHP consisting of 15 turns with copper material and the diameter of tube 0.1375cms respectively. Also working fluids are acetone, methanol and deionized water and the filling ratios between 30-70%. Heat inputs are supplied to the evaporator side of the pulsating heat pipe varied from 80 to 180 W. The experimental results proved that Methanol serves as a better working fluid compared to acetone and ethanol with fill ratio of 60%.

There are mainly two types of pulsating heat pipe, open loop and closed loop. In open loop PHP the condenser region is open so the water and the water vapour that rises from the evaporator due to heat flux imposed in the later leaves the PHP from one of the end. Here the continuous supply of water, the working fluid, is needed. However in the closed loop PHP there is a continuous circulating of working fluid within the PHP over the whole time.

There is an obvious gap in the published literature on CFD simulations of the two phase heat transfer/flow within a wickless heat pipe. Therefore, the purpose of this paper is to build a CFD model to cover all the details of two-phase flow and heat transfer phenomena during the operation of a wickless heat pipe charged with Acetone. The developed CFD model has been validated by experimental paper [1] with good agreement.

2 GOVERNING Equations

The general approach to the modeling of two phase flow using a single fluid formulation based on VOF method will be followed for modeling the slug flow in capillaries. The hydrodynamics can be described by the equations for the conservation of mass, momentum and energy together with an additional advection equation to determine the gas-liquid interface, called Volume of Fluid (VOF). The formulations of the governing equation are as follows:

Conservation of Mass:

$$\nabla.\vec{u} = 0 \tag{1}$$

Conservation of Momentum:

$$\frac{\partial(\rho\vec{u})}{\partial t} + \nabla.(\vec{u}.\rho\vec{u}) = -\nabla p + \nabla.[\mu(\nabla\vec{u} + (\nabla\vec{u})^T)] + \overrightarrow{f_\sigma} \tag{2}$$

Conservation of Energy:

$$\frac{\partial(\rho cT)}{\partial t} + \nabla.(\rho c\vec{u}T) = \nabla.(k\nabla T) \tag{3}$$

Where ρ is the density, $\vec{u}$ is the velocity vector, p is the pressure,μ is the dynamic viscosity, c is the specific heat capacity, T is the temperature and k is thermal conductivity. The source term $\overrightarrow{f_\sigma}$ in above momentum equation represents surface tension force. The governing equations are solved in the phases and interface is represented by the volume of fluid (VOF) method.

3 CFD SIMULATION

The main objective is to study the heat transfer performance of a PHP with varying load conditions in evaporator using acetone as working fluid. This study is done in CFD software ANSYS.

Also the flow pattern of acetone and acetone vapor within the PHP is to be observed so that the path can be predicted. Moreover, the temperature variations in evaporator and condenser can be referred as the performance parameter.

3.1 Methodology

First of all the literature survey is done on the operation and performance of pulsating heat pipe. Relevant recommendations made on the papers and journals are considered to come up with the defined problem statement. In this regard, the paper by Dharma pal A Baitule et. al. [1] is taken as the reference paper for the basic geometry, operating conditions and performance parameter. This paper is relied on the experimental analysis and has left the ground for the CFD analysis. Later the tools for solving the model is decided and worked on. After setting suitable schemes, simulation is run till the desired convergence criteria are achieved. There are many tools and techniques using which results can be displayed. These data from the result are categorized and then compared with our basic reference. The result of computer based CFD is then analytically verified.

3.2 Geometry and Meshing

In this work for creating geometry, the reference of geometry design of PHP is taken into consideration as suggested by Dharma pal A Baitule et. al. [1] the grid (mesh) independent study is carried out for every model that is used and made sure that the solution is also independent of the mesh resolution.

3.2.1 Geometry

PHP is a closed type with Single turn of copper tube which has inner and outer diameter 1.95 mm and 3mm respectively. The copper domain is not considered in the geometry modeling, as only the fluid domain is the matter of consideration. The created geometry is shown in fig. 1.

3.2.2 Meshing

In addition to the automated settings, ANSYS Meshing provides additional control with the option to specify

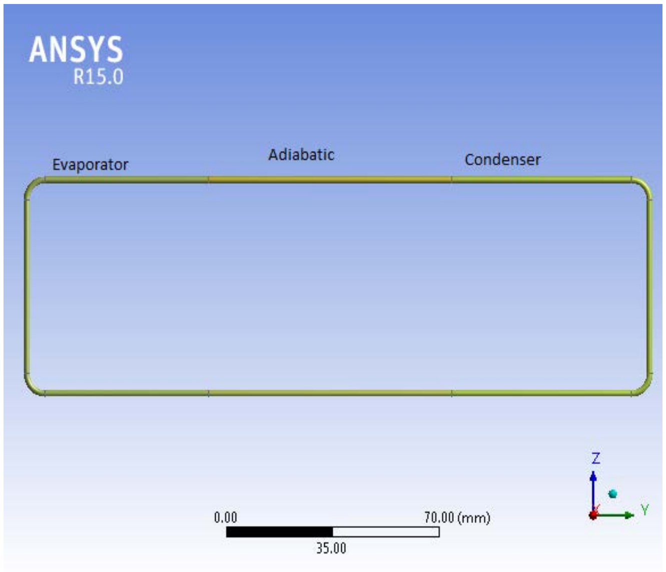

Fig.1 Geometry modelling of a single turn php

combinations of point controls, edge controls, surface controls and/or body controls. Each one of these has its own options and can be used to influence the mesh in different ways. In this case, the automatic method for mesh shape is selected; however the sizing for the mesh is done manually. The minimum and maximum mesh sizes are chosen to be 0.0002mm as shown in fig. 2. With this control setting, meshing is generated with 6, 64,427 number of nodes and 6, 17,874 number of elements. It is the meshing section where the sections of the geometry are given name which makes easy while defining domain and giving boundary conditions in the set up section of CFX-PRE.

3.3 Analysis Setting

Steady state analysis is done on CFX in the first phase. Though most of the thermal analysis with phase change phenomenon is mainly a transient case, steady state analysis is run just to be sure that this case is of steady state analysis or the transient. The difference between a steady state simulation and marching a transient solution to steady state is that the steady state simulation ignores many of the cross terms and higher order terms dealing with time. These terms all go to zero in steady state so they don't affect the steady state result. The transient simulation includes all these terms. Usually this means the steady state model has an easier convergence as there are fewer terms to model and some transient non-linearity are removed, but in a few models these non-linearity help convergence (but this is infrequent).

Fig.2 Meshing in ANSYS CFX

While performing the simulation several combinations of settings were tested. Parameters were systematically changed in order to investigate their effect on the results. In the first part, as mentioned earlier steady state analysis was performed and later the simulation is switched to transient state. The effect of dispersed phase diameter, turbulence model, and drag law, discretizing scheme for the gas volume fraction (GVF) equation, phase formulation and turbulence dispersion force was investigated. In the latter part, focus was on investigating the use free surface model. In addition to the parameters previously mentioned, the effect of mesh size and steady state versus transient time formulation was investigated. In meshing also, coarser and refined meshes were investigated to find the influence of mesh size. Regarding time formulation, a steady state simulation is much less time consuming than a transient simulation, however, multiphase flows often exhibit transient behavior and forcing a transient flow in to a steady stat might produce an unphysical solution. A transient simulation was therefore run for the final investigation approach to investigate the transient behavior.

For the transient state analysis different approach of simulations, viz. time steps, time steps for run and adaptive methods were done as a trail. Later the time steps option is choose to proceed with. Given the number of meshes and the flow velocity of the fluid in the domain, larger time steps could not be used. This confines the simulation to be done with smaller time step size. No external solver coupling was done as there is no need of that, since all the parameters, ranging from geometry, meshing to the domain settings was performed under the same file; and the chosen solver i.e. CFX is capable in solving the very case on its own.

3.4 Basic Settings for Domains

Mainly three domains were defined for the overall geometry of PHP, viz. condenser, adiabatic region and evaporator. Moreover, the adiabatic region is divided into two domains so as to incorporate the acetone level rise in the PHP from the evaporator at the base. For water ratio fill of 60% by volume, the acetone from the adiabatic region's base.

3.5 Boundary Conditions

Different heat fluxes were given to Evaporator region, they are 9W, 11W, 13W and 15W.For Adiabatic Region Heat flux is zero. In condenser region negative heat flux was given, this heat flux is calculated from experimental reference paper.VOF model is used for the CFD analysis. In all domains, the walls are of no slip wall types. All the walls are assumed smooth walls. In condenser the thermal boundary condition with fixed temperature of 302K is set. Similarly, in adiabatic regions the boundary condition is adiabatic which defines the heat flux rate as 0 Wm^-2K^-1. Likewise in evaporator region, where the external heat source is attached, total constant heat flux of value 7945 Wm^-2K^-1.

3.6 Turbulence Model

One of the most prominent turbulence models, the k-epsilon model, has been implemented in this analysis as is done in most of the general purpose CFD codes and is considered the industry standard model. It has proven to be stable and numerically robust and has a well-established regime of predictive capability. For general purpose simulations, the k-epsilon offers a good compromise in terms of accuracy and robustness.Within CFX the k-epsilon turbulence model applies standard two-equation models. These two equations are nothing but the transport equations (or partial differential equations) each for turbulent kinetic energy (k) and dissipation (ε) and uses the scalable wall-function approach to improve robustness and accuracy when the near-wall mesh is very fine. Thus it provides good predictions for the flow of fluids in the domain.

3.7 Condenser

In condenser region, all components of velocity (u, v and w) are provided as 0 m/s and relative pressure, 0 kPa. Assuming that there will be less turbulent flow in the condenser, as there is only low pressure air at low temperature in the initial state. As per the problem statement the volume fraction value for air is 1, and acetone or acetone vapor both are set to 0 volume fraction.This is because there is no acetone or acetone vapor in the condenser in the first place, and after the heating of evaporator is heated, if the acetone or acetone thermal boundary condition with fixed temperature of 302K is set.

3.8 Evaporator

In evaporator also all the components of velocity is set to 0 m/s and relative pressure to 0 kPa. As far as volume fractions are concerned values of 1, 0 and 0 is provided for acetone, acetone vapor and air respectively. In all domains, the walls are of no slip wall types. All the walls are assumed smooth walls. Similarly, in adiabatic regions the boundary condition is adiabatic which defines the heat flux rate as 0 Wm^-2K^-1. Likewise in evaporator region, where the external heat source is attached, total constant heat flux of value 7949 Wm^-2K^-1.

4 Results and Discussions

The output of the simulation can be observed in many ways like contours, streamlines, chart or graphs. The values of various parameters over the time steps are noted (as shown in table 1) and they are plotted to produce graphs. Acetone temperature at evaporator and condenser are plotted, and also volume fractions of Acetone and Acetone vapour at condenser and evaporator are plotted so as to get the clearer picture of the output. By observing these graphs, from fig. 3 to fig.10 the results of the simulation can be analyzed. Also, by observing fig. 4, that validation of CFD results with Experimental data is achieved.

Heat Flux (W)	Evaporator Temperature(K)	Condenser Temperature(K)
9	312	292
11	316	292.5
13	320	293
15	324	294

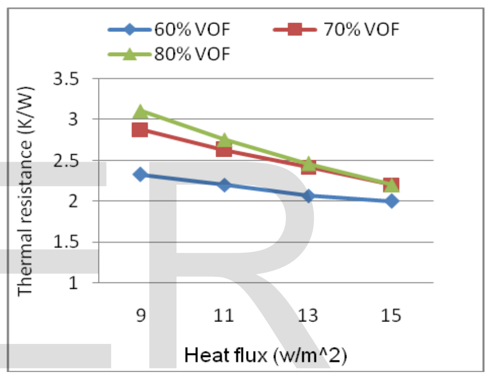

Fig.3 Comparison graph for variation in Thermal Resistance with heat flux at different volume fractions.

It is inferred from the above fig. 3 is that the Thermal resistance is low at 60% volume fraction as compared with 70% and 80% volume fractions. Thus acetone with 60% volume fraction shows the better result i.e. decrease in the thermal resistance is more with respect to heat flux respectively.

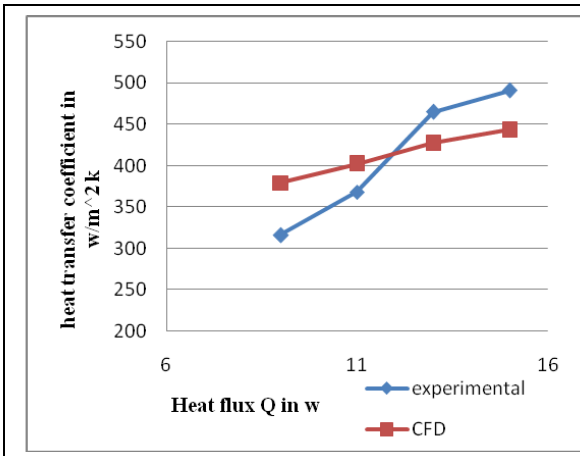

Fig.4 Comparison graph between Experimental and CFD results for variation in heat transfer coefficient with heat flux at 60% volume fraction.

fig. 4 is that the heat transfer coefficient is increasing with increase in heat flux. The maximum deviation observed between experimental and CFD analysis is 16%. And the deviation is decreasing with increase in heat flux value.

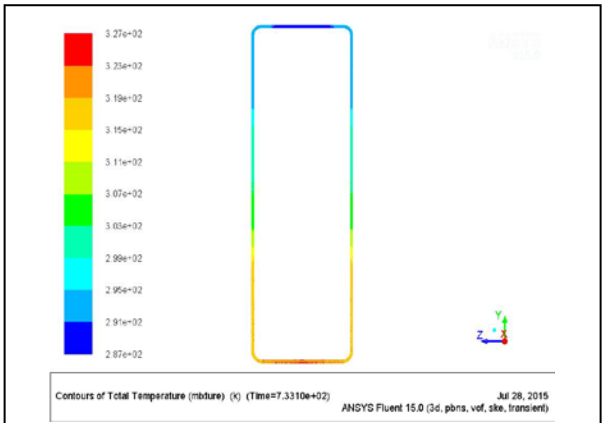

Fig.5 Temperature Contours of PHP at 9W and 60% volume fraction for acetone working fluid.

4.1 Operational Extremities

For 0% filling ratio, the PHP runs without any fluid inside it and this serves as the reference Measurement. Obviously the mode of heat transfer is by pure conduction in PHP. The other extreme is when the PHP is fully filled with the working fluid (FR = 100%). In this mode the heat transfer is due to the single-phase buoyancy induced liquid circulation in the PHP.

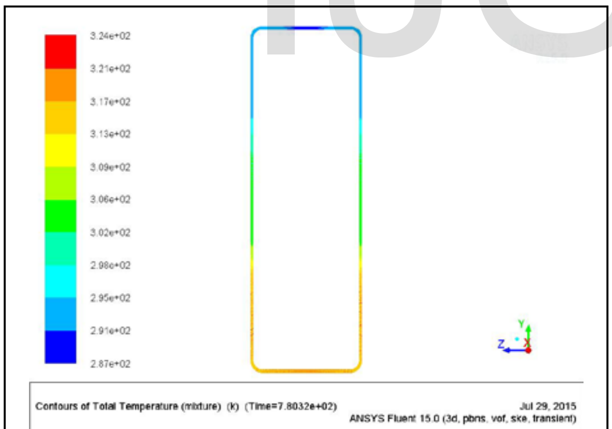

Fig.6 Temperature Contours of PHP at 9W and 70% volume fraction for acetone working fluid.

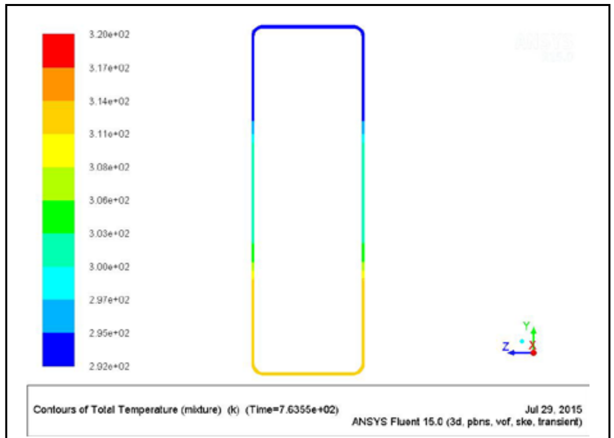

Fig.7 Temperature Contours of PHP at 9W and 80% volume fraction for acetone working fluid.

In this case the local heat transfer coefficient in the tubes is only a function of fluid Grashof and Prandtl numbers. There is a smooth decrease of the thermal resistance with increasing heat power input. Similarly low input heat fluxes were not capable of generating enough perturbations and the resulting bubble pumping action was extremely restricted. Over all, this scenario results in poor performance (i.e., very high thermal resistance). As the heat input was increased, it improved the heat transfer coefficient to marked degree. Still higher input heat fluxes resulted in bulk flow taking a fixed direction that did not reverse with time. This circulation was manifested as adjacent tube becoming alternately hot and cold. Interestingly, in such a case lowest thermal resistance was observed. For 9W heat input with acetone as working fluid and for different fill ratios is observed in fig.5, fig.6 and fig.7.

4.2 Effect of Filling Ratio on Performance Limit

The filling ratio is defined as the fraction by volume of heat pipe, which is initially filled with the liquid. The optimum filling ratio is determined experimentally [1] when the maximum heat transfer rate is achieved at the given temperature. As we know that below FR 10%, there is no enough working fluid in the system for substantial sensible and latent heat transfer. The evaporator trends to dry out. The heat transfer performance is poor with high thermal resistance and low heat transfer limit. 80% there are not enough. Above FR bubbles to provide the pumping action, so the thermal performance drastically deteriorates. Between about 20% to 80% fill ratio, the PHP functions is in a true pulsating mode, and the thermal resistance is clearly lower than that of 100% filled mode. Otherwise, the heat transfer limit increases with the filling ratio. Similarly if the heat input increased from lower to higher, the fluctuation of thermal resistance is continuously decreased. At 40% and 80% filling of PHP give the higher thermal Resistance. The optimum fill ratio for the PHP in the experiments [1] is about 60%, which most adequately combines the advantages of the following two aspects: the latent heat along with the pumping action of the bubbles, and the sensible heat transport of the liquid slugs. For 15W heat input with acetone working fluid and for different fill ratios of acetone is ob-

ISSN 2229-5518
served in fig.8, fig.9 and fig.10.

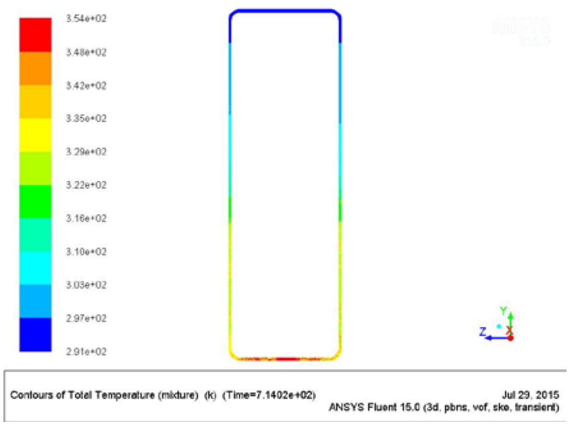

Fig.8 Temperature Contours of PHP at 15W and 60% volume fraction for acetone working fluid.

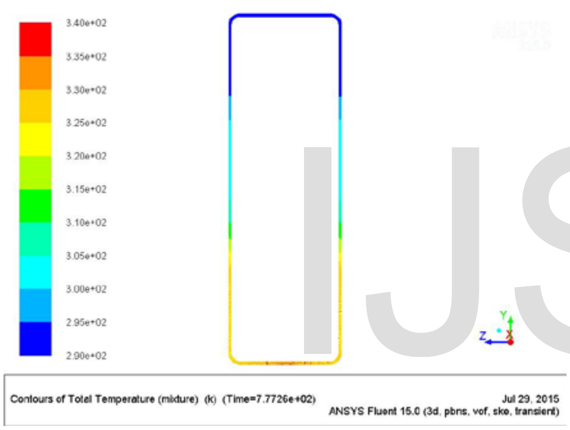

Fig.9 Temperature Contours of PHP at 15W and 70% volume fraction for acetone working fluid.

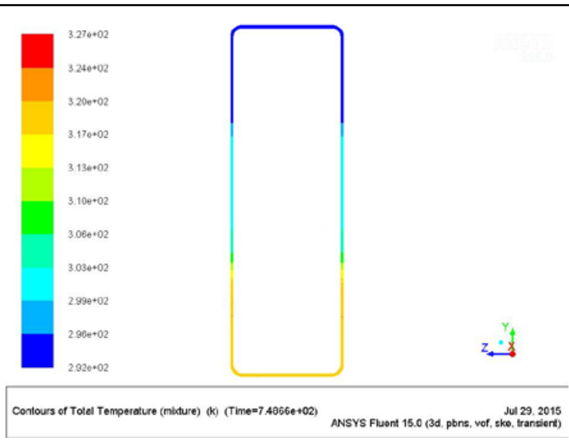

Fig.10 Temperature Contours of PHP at 15W and 80% volume fraction for acetone working fluid.

It observed from these volume fraction contours is as time step size increases the volume fraction of acetone-liquid decreases when heat load given at evaporator section. And also observed that the acetone-liquid volume fraction decreases as heat load increases in the PHP. This is happens because the heat load is increases at the evaporator section then the acetone-liquid to vapour phase transformation time is reduced

5 CONCLUSIONS

•In this research based on CFD analysis of PHP it can be concluded that two phase flow is successfully simulated in CFX.

•In CFD analysis the variation in evaporator and condenser wall temperatures with flow time is observed

•It is observed that the thermal resistance decreases and heat transfer coefficient increases with increase in heat flux due to chaotic fluid movement.

•And among three fill ratios 60%, 70%, 80% of acetone, 60% is observed to be more suitable Fill ratio for PHP operation under different operating conditions.

•Thus at a fill ratio of 60%, the PHP is exhibiting better heat transfer charecteristics.

REFERENCES

[1] Naik, Rudra, et al. "Experimental investigation and performance evaluation of a closed loop pulsating heat pipe." Journal of Applied Fluid Mechanics 6.2 (2013): 267-275.

[2] Qu, Jian, Hui-Ying Wu, and Qian Wang. "Experimental investigation of silicon-based micro-pulsating heat pipe for Cooling Electronics." Nanoscale and Microscale Thermophysical Engineering 16.1 (2012): 37-49.

[3] Shafii, M. B., et al. "Experimental investigation of pulsating heat pipes and a proposed correlation." Heat Transfer Engineering 31.10 (2010): 854-861.

[4] Clement, Jason, and Xia Wang. "Experimental investigation of pulsating heat pipe performance with regard to fuel cell cooling application." Applied Thermal Engineering 50.1 (2013): 268-274.

[5] Pachghare, Pramod R., and Ashish M. Mahalle. "Effect of pure and binary fluids on closed loop pulsating heat pipe thermal performance." Procedia Engineering 51 (2013): 624-629.

[6] Taslimifar, Mehdi, et al. "Overall thermal performance of ferrofluidic open loop pulsating heat pipes: An experimental approach." International Journal of Thermal Sciences 65 (2013): 234-241.

[7] Fadhl, Bandar, Luiz C. Wrobel, and Hussam Jouhara. "CFD modelling of a two-phase closed thermosyphon charged with R134a and R404a." Applied Thermal Engineering 78 (2015): 482-490.

ISSN 2229-5518

[8] Baitule, Dharmapal A., and Pramod R. Pachghare. "Experimental Analysis of Closed loop Pulsating Heat Pipe with variable Filling ratio." International Journal of Mechanical Engineering & Robotic Research 2.3 (2013).

[9] Naik, Rudra, et al. "Experimental investigation and performance evaluation of a closed loop pulsating heat pipe." Journal of Applied Fluid Mechanics 6.2 (2013): 267-275.

[10] Rudresha, S., and Vijee Kumar. "CFD Analysis and Experimental Investigation on Thermal Performance of Closed loop Pulsating Heat pipe using different Nanofluids." International Journal of Advanced Research 2.8 (2014): 753-760.