Transilvania
University
of Brasov
FACULTY OF
MECHANICAL ENGINEERING

COMAT 2020 & eMECH 2020

Brasov, ROMANIA, 29-31 October 2020

NUMERICAL STUDY OF BOILING FLOW IN A HEAT PIPE

V.B. Ungureanu[1], L. Costiuc[1]
[1] Transilvania University of Brașov, ROMANIA, virbung@unitbv.ro, lcostiuc@unitbv.ro

Abstract: *An investigation to determining the working fluid for providing passive air-side cooling was carried out. The effectiveness of water as heat pipe working fluid was analyzed by computing the thermal cooling capacity under a low Reynolds number, usually found under natural ventilation. The internal multiphase flow profiles and the external air temperatures were numerically predicted using Computational Fluid Dynamics (CFD). The present work intends to participate to the sustainable operation of replacing synthetic refrigerants with a natural fluid, such as water, in delivering energy-free cooling.*

Keywords: *numerical simulation, heat pipe, heat transfer, CFD*

1. INTRODUCTION

The numerical and analytical simulation of heat pipes has advanced significantly in the last years. The state of the art models has been advanced in steady state, transient simulation for heat pipes of conventional and non-conventional geometries.

Because the determining of the heat capacity transmitted through a heat pipe or heat pipe performance characteristics is the main goal, it is necessary to determine the liquid and vapor pressure losses in the separate regions. The phenomena occurring within a heat pipe, according to Fig. 1, can be divided into next categories: (1) vapor flow in the core region, (2) liquid flow in the wick, (3) interaction between the liquid and vapor flow and (4) heat conduction in the wall. Most of the analytical and numerical research on heat pipes has been done for the vapor core region and wall heat conduction, because the liquid flow is difficult to describe with an exact theoretical model. The recent models for porous fluid flow may increase the performance of this modeling. The presence of the wick structure requires some empirical information that must be obtained from experiments.

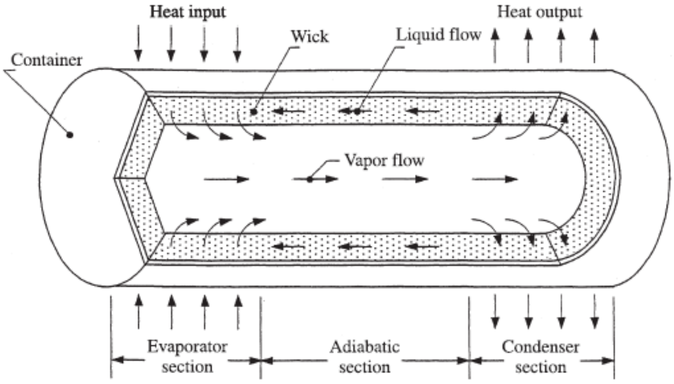

Figure 1. The main regions of the heat pipe

2. NUMERICAL HEAT TRANSFER MODELING

The numeric model was developed using COMSOL Multiphysics software in a laminar convection for vapor flow. The model is considered as 3D steady-state with surface heat load at the evaporator and convection heat transfer at the condenser using the equations for non-isothermal flow, Laminar Fluid-Thermal interface for convective and conductive heat transfer combined with evaporation/condensation mechanism modeling.

The main assumptions for this model are: the gravity effects are neglected, the heat transport takes place mainly by evaporation/condensation and convection of vapor, the heat transport in the wick is simplified as a conduction with an effective heat transfer coefficient and the constant material properties are assumed, except for the vapor density.

2.1. Geometry

The 3D model has been created in Comsol Multiphysics consisting of three block type objects, representing the metal material of the heat pipe, the wick and the vapor zone. Because of symmetry the half of the geometry is presented in Fig. 2.

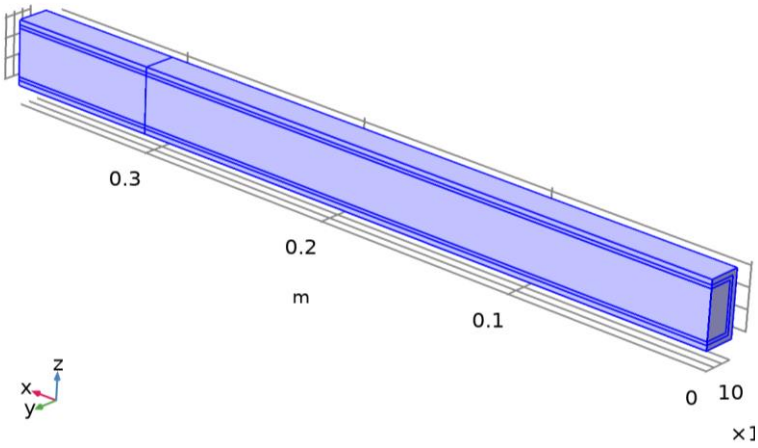

Figure 2. The geometry of the heat pipe

The length of heat pipe is L=0.375 m, width = height = 32 mm, wall = 2.5 mm, and wick width = 1.65 mm. The length of heating zone, corresponding to the evaporator was considered as Lq = 75 mm.

2.2. Geometry-Mesh.

The 3D heat transfer and vapor flow model after the discretization using mesh elements is shown in Fig. 3, and is resulting 726719 elements (726083 tetrahedra + pyramids+prisms), with minimum element quality of 0.1544.

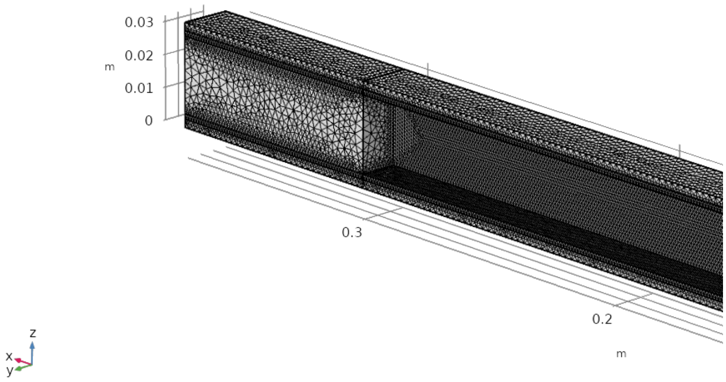

Figure 3. The mesh of the heat pipe

2.3. Model set-up

The boundary conditions at inlet and outlet were set up as: a heat flux inlet on evaporator, and a temperature and convective heat transfer on condenser. The boundary conditions are presented in Table 1.

Table 1 Entry data for the model

Property	Value
Latent heat of evaporation/condensation	2473[kJ/kg]
Reference temperature T_ref	100[degC]
Reference pressure p_ref	1[atm]
Heating flux Q_in	500[W]
Convective heat transfer coefficient /condenser/adiabatic zone	1200[W/m^2.K]
Ambient temperature	285 [K]
Initial heat pipe temperature	293.15[K]

3. RESULTS

The results obtained with the 3D model described above are presented below.
The temperature distribution in HP channel in surface plot is presented in Fig.4, and the isothermal isosurfaces is presented in Fig.5. In the Fig.5 the white interrupted zone shows the adiabatic zone of the heat pipe where the heat flux with ambient is zero.

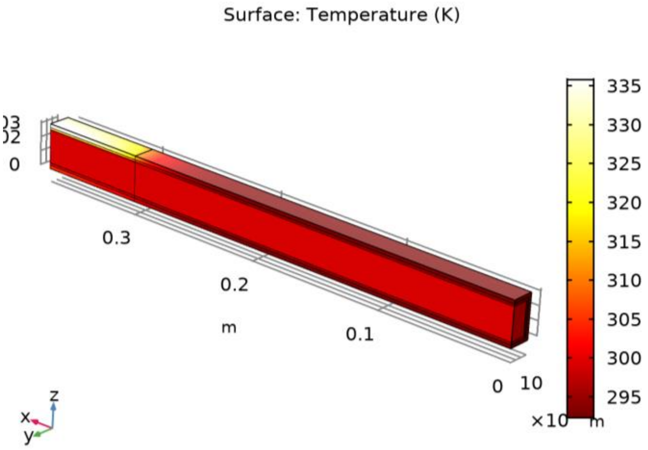

Figure 4. The heat pipe surface temperature [K]

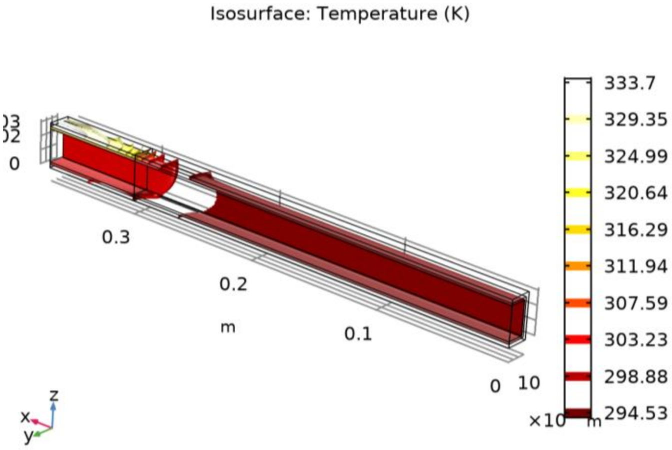

Figure 5. The heat pipe isothermal contours[K]

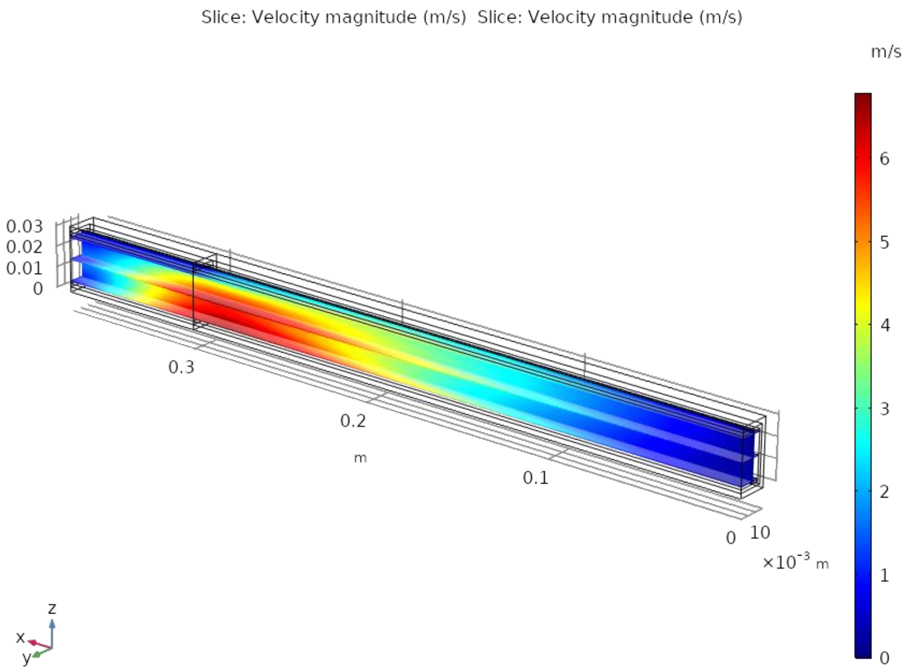

Figure 6. The velocity of vapors into the heat pipe [m/s]

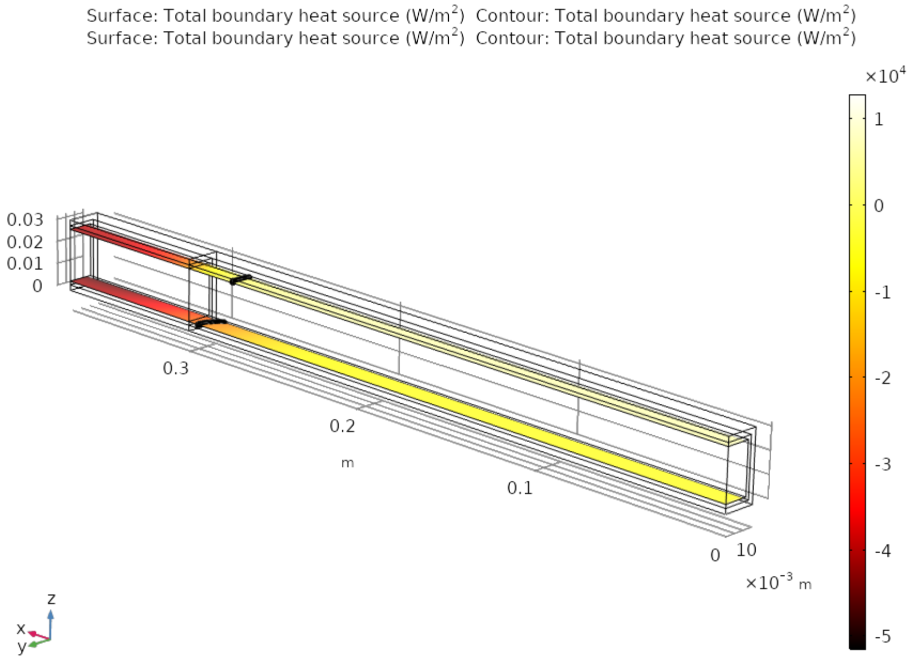

Figure 7. The specific heat flux on evaporator and condenser boundary [W/m^2]

The velocity field is presented in Fig. 6. The boundary heat flux distribution, negative for the evaporator and positive for the condenser, is presented in Fig. 7.

4. CONCLUSION

The basic heat pipe consists of a vacuum chamber from which all non-condensable gases have been discharged and which contains a capillary structure and a certain amount of two-phase fluid. Thus, the heat pipe is a device that achieves an efficient heat transfer by combining in a closed cycle the phenomena of vaporization, vapor transport, condensation and condensation return, of a working fluid. At first glance, the operation of the heat pipe seems to be extremely simple. The thermal energy is transported from the evaporator to the condenser through a continuous cycle of mass transfer and phase change of a working fluid. The CFD model presented here assured the main processes that are involved in functioning of heat pipes.

REFERENCES

[1] Faghri, A., 2012, "Review and Advances in Heat Pipe Science and Technology," Journal of Heat Transfer, 134(12), 123001. http://dx.doi.org/10.1115/1.4007407

[2] Faghri, A., 1995, Heat Pipe Science and Technology, 1st ed., Taylor & Francis, Washington, D.C.

[3] Faghri, A., and Thomas, S., 1989, "Performance Characteristics of a Concentric Annular Heat Pipe: Part I-Experimental Prediction and Analysis of the Capillary Limit," Journal of Heat Transfer, 111(1-4), 844-850. http://dx.doi.org/10.1115/1.3250795

[4] Faghri, A., Chen, M. M., and Morgan, M., 1989a, "Heat Transfer Characteristics in Two-Phase Closed Conventional and Concentric Annular Thermosyphons," Journal of Heat Transfer, 111(3), 611-618. http://dx.doi.org/10.1115/1.3250726

[5] Fetcu ,Ungureanu, 1999, Tuburi termice, Ed. Lux Libris