Applied Thermal Engineering 60 (2013) 122–131

Contents lists available at SciVerse ScienceDirect

Applied Thermal Engineering

journal homepage: www.elsevier.com/locate/apthermeng

Numerical modelling of the temperature distribution in a two-phase closed thermosyphon

Bandar Fadhl [1], Luiz C. Wrobel*, Hussam Jouhara

School of Engineering and Design, Brunel University, Kingston Lane, Uxbridge, Middlesex UB8 3PH, UK

HIGHLIGHTS

- CFD modelling of two-phase flow and heat transfer in a wickless heat pipe.
- Evaporation, condensation and phase change included by user-defined functions.
- Simulation results validated by experimental measurements.
- Simulation was successful in reproducing heat and mass transfer processes.
- Good agreement observed between CFD and experimental temperature data.

ARTICLE INFO

Article history:
Received 29 April 2013
Accepted 24 June 2013
Available online 2 July 2013

Keywords:
Thermosyphon
Computational fluid dynamics (CFD)
Multiphase flow
Phase change material
Evaporation
Condensation

ABSTRACT

Interest in the use of heat pipe technology for heat recovery and energy saving in a vast range of engineering applications has been on the rise in recent years. Heat pipes are playing a more important role in many industrial applications, particularly in improving the thermal performance of heat exchangers and increasing energy savings in applications with commercial use. In this paper, a comprehensive CFD modelling was built to simulate the details of the two-phase flow and heat transfer phenomena during the operation of a wickless heat pipe or thermosyphon, that otherwise could not be visualised by empirical or experimental work. Water was used as the working fluid. The volume of the fluid (VOF) model in ANSYS FLUENT was used for the simulation. The evaporation, condensation and phase change processes in a thermosyphon were dealt with by adding a user-defined function (UDF) to the FLUENT code. The simulation results were compared with experimental measurements at the same condition. The simulation was successful in reproducing the heat and mass transfer processes in a thermosyphon. Good agreement was observed between CFD predicted temperature profiles and experimental temperature data.

1. Introduction

A heat pipe is a two-phase heat transfer device with a highly effective heat transfer rate through evaporating and condensing a fluid that is circulating in a sealed container. A wickless heat pipe, or a two-phase closed thermosyphon, relies on gravitational forces to return the working fluid to the evaporator. This is different from a wicked heat pipe, where the working fluid is returned from the condenser by capillary forces [1–4]. Heat pipes have been successfully used for waste heat energy recovery in a vast range of engineering applications, such as heating, ventilation, and air conditioning (HVAC) systems [5], ground source heat pumps [6], water heating systems [7] and electronics thermal management [8]. This is mainly because of their simple structure, special flexibility, high efficiency, good compactness, and excellent reversibility [9–12]. Thermosyphons have three sections, which are the evaporator at the bottom end, where heat is added and the liquid is vaporised, the condenser at the top end, where heat is released and the vapour is condensed, and an adiabatic section in the middle between the evaporator and condenser [13].

In a thermosyphon, heat is added to the evaporator where a liquid pool exists, changing the liquid into vapour. The high temperature and pressure cause the vapour to flow and pass through the adiabatic section towards the condenser. The vapour adjacent to the condenser's wall gives up its latent heat that is absorbed in the evaporator section. The condensed liquid is then transported back to the evaporator due to gravity [14].

* Corresponding author. Tel.: +44 1895266696.
E-mail address: luiz.wrobel@brunel.ac.uk (L.C. Wrobel).
[1] On leave from Department of Mechanical Engineering, Umm Al-Qura University, Makkah, Saudi Arabia.

1359-4311/$ – see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.applthermaleng.2013.06.044

Two-phase closed thermosyphons have been extensively used in many applications [15]. However, only a limited number of CFD numerical simulation studies on two-phase closed thermosyphons have been published. Alizadehdakhel et al. [1] provided a two-dimensional model and experimental studies in which they investigated the effect of input heat flow and fill ratio of the working fluid on the performance of a two-phase closed thermosyphon. They validated their study using experimental results. Three input heat flow rates of 700, 500, and 350 W and three fill ratios of 0.3, 0.5, and 0.8 were considered. Under these operating conditions, they found the performance of the thermosyphon improved when the input heat flow was increased from 350 to 500 W. Further, they discovered the best performance was at a fill ratio of 0.5. The authors reported a term called “heat performance”, which they calculated by using the following equation for different fill ratios:

$$\eta = \frac{Q_{out}}{Q_{in}} \times 100$$

However, this term is not usual in heat pipe publications. In general, the thermal performance term used to characterize thermodynamics at different heat throughputs is the total thermal resistance.

Legierski et al. [14] provided CFD modelling and experimental measurements of heat and mass transfer in a horizontal wicked heat pipe. They investigated the effectiveness of the heat pipe thermal conductivity in a transient state during start-up of the pipe operation and during temperature increases. The authors used a heat pipe that was 200 mm long with 4 mm diameter and 25 mm length for the evaporator and condenser. They also used two containers, one for hot water (90 °C) at the evaporator section and one for cold water (ambient temperature) at the condenser section. They developed a three-dimensional CFD model to simulate the internal vapour flow. They found that the effective thermal conductivity of the wicked heat pipe depended on the time in the range between 15×10^3 and 30×10^3 W/m K, and achieved its steady-state value after approximately 20–30 s. However, the authors did not consider in the CFD modelling the phase change material from liquid phase to vapour phase, as well as condensation in the condenser section and pool boiling in the evaporator section.

Zhang et al. [16] developed a two-dimensional heat and mass transfer model for a disk-shaped flat two-phase thermosyphon used in electronics cooling. The authors simulated the vapour flow inside the flat two-phase thermosyphon as a single-phase flow. They compared their predicted model with experimental results to determine the factors that affected the axial thermal resistance of a thermosyphon. This model was limited as it considered the flow inside the flat thermosyphon as a pure vapour phase only.

Joudi and Al-Tabbakh [17] numerically studied a two-phase thermosyphon solar domestic hot water system, by using a computer simulation. They used R-11 as a working fluid in the thermosyphon. Firstly, the authors validated the computer program and calculation procedure by comparing the results with those obtained with single-phase systems. They then performed calculations for the two-phase thermosyphon system. In their calculations, they evaluated mass flow rate, saturation pressure, and temperature in the collector and condenser, together with tank temperature and collector and condenser thermal efficiencies. The results of the study showed that the collector efficiency of the two-phase system was approximately 20% greater than in a single-phase system. Further, the response of the two-phase system in reaching maximum tank temperature and efficiency was faster than in a single phase system. This study was only a mathematical model and did not include any flow visualisation.

Annamalai and Ramalingam [18] carried out an experimental investigation and CFD analysis of a wicked heat pipe using ANSYS CFX. The authors considered the region inside the heat pipe as a single phase of vapour and a wick region as the liquid phase. They compared the predicted surface temperature along the evaporator and condenser walls and the vapour temperature inside the heat pipe with the experimental data. This model treated the flow inside the heat pipe as a single-phase and did not include the evaporation, condensation and phase change processes.

De Schepper et al. [19] developed a model to simulate the evaporation process of a hydrocarbon feedstock in a heat exchanger. They used the VOF and UDF techniques to simulate flow boiling including the phase change process. They proposed correlations to calculate the mass and heat transfer between the phases that were able to simulate the evaporation and boiling phenomena inside the convection section of a steam cracker. This model was for the convection section in a steam-cracking furnace; however, it did not include the heat pipe system.

Lin et al. [20] built a CFD model to predict the heat transfer capability of miniature oscillating heat pipes. The effects of different heat transfer lengths and inner diameters at different heat inputs were used to analyse the heat transfer capability of MOHPs. They compared the predicted model with experimental results. This model did not visualise the internal phenomena of evaporation, condensation and phase change inside the MOHPs.

Heat pipe technology is currently still under development. However, there are limited studies on the validation of predictions for modelling closed two-phase thermosyphons or wickless heat pipes. Further, a CFD simulation of a wickless heat pipe that considers all the details of heat transfer phenomena inside the heat pipe has not yet been reported. Hence, a gap still exists for further CFD work to model a wickless heat pipe. Additionally, CFD models can reduce the amount of experimental work. Therefore, in this paper, a comprehensive CFD modelling has been employed to cover all details of two-phase flow and heat transfer phenomena during the operation of a straight wickless heat pipe. Moreover, a user-defined function (UDF) has been used to complete the FLUENT code in order to simulate the phase change material.

2. Experimental apparatus

In order to validate the CFD findings, an experimental apparatus was built to carry out a thermal performance investigation on a typical wickless heat pipe.

The experimental apparatus used in the current investigation is shown in Fig. 1. The apparatus consists of a two-phase closed thermosyphon (TPCT), a rope heater, the cooling water circuit, and instrumentation. The apparatus was fixed on a framework to insure vertical orientation under all test conditions.

The TPCT was manufactured from a 22 mm outer diameter, 0.5 m-long smooth copper tube with a wall thickness of 0.9 mm. It consists of a 0.2 m-long evaporator section, a 0.1 m-long adiabatic section and a 0.2 m-long condenser section (see Fig. 1).

The evaporator section was heated by a rope heater with a maximum power output of 500 W at 220 V, which was evenly wrapped and not directly positioned above any of the thermocouples that are used to measure the surface temperature of this section. The energy output of the heater was controlled by a variac. The evaporator section was wrapped in a layer of fire-proof insulation before it was wrapped with suitable thermal insulation layers to minimise any heat losses to the ambient. The condenser section was cooled using a double pipe concentric heat exchanger with an insulated outer surface. The 10-cm long adiabatic section was also well insulated to ensure no heat energy interactions take place with the ambient. The insulated adiabatic section wall