International Journal of Heat and Fluid Flow 65 (2017) 177–191

Contents lists available at ScienceDirect

International Journal of Heat and Fluid Flow

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

# Volume of fluid approach of boiling flows in concentrated solar plants

R. Dinsenmeyer[a,b,c,*], J.F. Fourmigué[a,b], N. Caney[a,c], P. Marty[a,c]

[a] *Univ. Grenoble Alpes, F-38000 Grenoble, France*
[b] *CEA, LITEN, F-38000 Grenoble, France*
[c] *CNRS, LEGI, F-38000 Grenoble, France*

ARTICLE INFO

*Article history:*
Received 17 September 2015
Revised 12 April 2017
Accepted 30 April 2017

*Keywords:*
Two-phase flow
Boiling
CFD
Flow patterns
3D numerical simulation
Horizontal tube

ABSTRACT

The Computational Fluid Dynamics (CFD) model proposed in this paper allows the flow patterns that evolve during progressive boiling inside large scale horizontal tubes to be simulated from the initial vapor generation stage to large vapor slugs. The volume of fluid (VOF) model was employed in combination with relatively simple hypothesis. The aim of the present work is to improve the design of receiver tubes at concentrated solar power plants with direct steam generation by simulating the evolution of flow regimes within these tubes. Despite numerous studies conducted in the past years on convective boiling, only a few made use of the VOF model to simulate large flow regime transitions. This work presents a preliminary and relatively qualitative approach to address this problem. Heat and mass transfer at the tube inner wall and at the liquid-gas interface were solved with the additional transport of two scalars. One accounts for the enthalpy field and the other represents the dispersed vapor phase of the liquid. This new phase was created at the wall surface of the liquid phase and rises up to the liquid-vapor interface. Different phenomena linked to the boiling process were taken into account: vapor creation at the wall, its transport, recondensation and the creation of large structures. This model was validated with boiling flow in a bent tube at different mass flow rates and heat fluxes, which allowed us to observe the evolution of two-phase flow patterns. Finally, numerical simulation of direct steam generation inside a concentrated solar plant receiver clearly showed the apparition and evolution of various two-phase flow patterns.

## 1. Introduction

Concentrated solar power (CSP) is a promising candidate for substitution of conventional power generation technologies. Solar radiation is concentrated on a pipe using mirrors, which heats up a working fluid to high temperatures (about 600 K). This fluid evaporates and produces electricity via a steam turbine coupled to a generator. When associated with a thermal energy storage system, transients can be overcome and their global efficiency increases. Sunlight can be focused to a single point (in parabolic and tower systems) or a line (in parabolic-trough and linear Fresnel collectors), the latter of which was used in this study. Currently most commercial plants use synthetic oil as a working fluid. The next generation of plants aims to directly use pressurised water (Eck et al., 2003) with direct steam generation (DSG) technology. By eliminating heat exchangers and the heat transfer fluid (oil or molten salt) that needs to be regularly replaced, operating costs can be reduced (Hirsch et al., 2014).

Imposing a uniform, relatively low heat flux on the tube walls results in a succession of two-phase flow patterns. First, subcooled liquid enters the tube and is heated up to the saturation temperature. Then bubbles are created at the inner tube surface and flow up to the top of the tube due to buoyancy. Then these bubbles coalesce causing vapor stratification at a low vapor velocity. Added vapor due to the boiling process increases the vapor velocity creating waves at the liquid-vapor interface. Under some conditions, these waves rise to the top of the tube forming liquid plugs that disrupt flow. As the vapor volume and velocity increase, liquid can be distributed around the perimeter (Kandlikar, 1999). The evolution of two-phase flow patterns induced by this vaporisation process can lead to uncontrolled flow and heat transfer due to its inherent instabilities (Murphy and Kenneth May, 1982).

Simulating the evolution of these flow regimes at large scales is challenging, but necessary in order to predict the system response. Presently, such systems are mainly simulated using a one-dimensional, system-based approach. However the use of 3-dimensional (3D) computational fluid dynamics (CFD) could lead to significant advances in the comprehension and sizing of CSP systems. This numerical approach is widely used in the nuclear industry, but we have found very few studies applying this

---

[*] Corresponding author at: 281 rue de la Balme, 38950 Saint Martin le Vinoux, France.

*E-mail address:* remi.dinsenmeyer@gmail.com (R. Dinsenmeyer).

http://dx.doi.org/10.1016/j.ijheatfluidflow.2017.04.011
0142-727X/

## Nomenclature

| | |
|---|---|
| $a$ | acceleration m.s$^{-2}$ |
| $A$ | cross-section area m$^2$ |
| $c$ | wave complex velocity m.s$^{-1}$ |
| $C_p$ | specific heat J.kg$^{-1}$.K$^{-1}$ |
| $D$ | diameter m |
| $E$ | energy J |
| $g$ | gravity acceleration m.s$^{-2}$ |
| $h$ | enthalpy J.kg$^{-1}$ |
| $h_{lv}$ | latent heat J.kg$^{-1}$ |
| $i$ | imaginary unit |
| $J$ | superficial velocity m.s$^{-1}$ |
| $k$ | thermal conductivity W.m$^{-1}$.K$^{-1}$ |
| $K$ | coefficient (Kutateladze) |
| $l$ | height m |
| $L$ | length m |
| $m$ | mass kg |
| $p_n$ | percentage |
| $P$ | pressure Pa |
| Pr | Prandtl number |
| $q$ | heat flux W.m$^{-2}$ |
| $S$ | volumetric source term kg.m$^{-3}$.s$^{-1}$ or W.m$^{-3}$ |
| $t$ | time s |
| $T$ | temperature K |
| $v$ | velocity m.s$^{-1}$ |
| $V$ | volume m$^3$ |
| $z$ | axial position m |

*Greek symbols*

| | |
|---|---|
| $\alpha$ | volume fraction |
| $\eta$ | perturbation height |
| $\varphi$ | passive scalar |
| $\kappa$ | wave number m$^{-1}$ |
| $\mu$ | dynamic viscosity Pa.s |
| $\rho$ | density kg.m$^{-3}$ |
| $\sigma$ | surface tension N/m |

*Subscripts*

| | |
|---|---|
| $b$ | related to boiling |
| $crit$ | critical |
| $d$ | related to the dispersed vapor |
| $E$ | energy |
| $ext$ | external |
| $g$ | related to the gas |
| $I$ | imaginary |
| $int$ | internal |
| $interf$ | related to the interface |
| $l$ | liquid |
| $lim$ | limit |
| $m$ | related to the mixture |
| $nucl$ | related to nucleation |
| $recond$ | related to recondensation |
| $rel$ | relative |
| $R$ | real |
| $s$ | related to the solid |
| $sat$ | saturation |
| $v$ | related to the vapor |
| $t$ | turbulent |
| $w$ | related to the wall |

technique to CSP technology. Two recent studies deal with DSG in parabolic-trough solar collectors (Lobon et al., 2014a,b). The authors used STAR-CCM+ CFD software combined with a homogeneous approach in order to access the transient behaviour of CSP plants under variable inlet conditions. Their results were validated using experimental data from the DISS test facility (Spain) and were based on parabolic trough collectors. However, due to the homogeneous nature of their model, flow patterns associated with the boiling process were not accessible.

CFD computation of convective boiling flows remains a challenge. In order to simulate flow pattern evolution, first the dispersed bubbles that rise up through the large vapor structures must be addressed. Transport of a dispersed phase must be determined as well as the localisation of the liquid-vapor interface. Dispersed bubble flow patterns are well simulated using inhomogeneous two-phase models, solving a momentum and energy equation for each phase (Ustinenko et al., 2008; Krepper and Rzehak, 2011). These models were mostly developed in order to study the local parameters of a boiling, two-phase flow near a wall. Numerous studies have been carried out by the nuclear industry that allow simulation of both dispersed bubbles and large structures via inhomogeneous models in adiabatic or condensation conditions (Vallée et al., 2008; Strubelj et al., 2009; Das and Das, 2010; Höhne and Vallée, 2010; Coste et al., 2012; Hänsch et al., 2012). Large vapor structures can also be simulated using specific models that allow for interface tracking, like the volume of fluid (VOF) model, but this model is often used in its adiabatic form. A few authors have developed custom code to simulate boiling flow (Yang et al., 2008; De Schepper et al., 2009). In these articles, a vapor creation calculation is determined from a constant based on experimental data. The combination of these two types of models is an interesting approach: interface tracking is necessary in order to simulate flow patterns with high void fractions while a dispersed model is sufficient to account for dispersed flow. A combination of the VOF model with a dispersed inhomogeneous model was applied to a 2-dimensional, small-scale adiabatic test case (Cerne et al., 2001).

This paper presents a CFD model that allows for the simulation of 3D flow pattern evolution in a heated tube during the boiling process and couples heat and mass transfer modeling at the tube wall. The VOF model is employed despite its limitations because it allows for satisfactory liquid-vapor interface simulation. An adiabatic validation of the VOF model for slug flow pattern simulation, followed by the presentation of our model with its preliminary and qualitative validation are shown. Finally, our model was applied to a real case of boiling inside a concentrated solar plant collector tube.

## 2. Adiabatic VOF model validation

### 2.1. Basic transport equations

To accurately predict the two-phase flow pattern evolution, a clear detection of the interface between the liquid and vapor is necessary. In order to track the interface and save computational resources, the VOF multiphase model was carried out using Ansys Fluent 14.5 software. In this model, a single set of conservation equations for mass and momentum were solved in transient regime for the mixture of the two phases:

$$\frac{\partial \rho_m}{\partial t} + \nabla \cdot (\rho_m \vec{v}_m) = S_m \tag{1}$$

$$\frac{\partial (\rho_m \vec{v}_m)}{\partial t} + \nabla \cdot (\rho_m \vec{v}_m \vec{v}_m) = -\nabla p + \nabla \cdot \left[\mu_m \left(\nabla \vec{v}_m + \nabla \vec{v}_m^{\,t}\right)\right] + \rho_m \vec{g} \tag{2}$$

An additional transport equation for the volume fraction was solved for the vapor phase only:

$$\frac{\partial \alpha_v \rho_v}{\partial t} + \nabla \cdot (\alpha_v \rho_v \vec{v}_v) = S_v \tag{3}$$