Condensation of steam on vertical wall

Zdeněk Sumara[1,*]

[1] ČVUT v Praze, Fakulta strojní, Ústav mechaniky tekutin a termodynamiky, Technická 4, 166 07 Praha 6, Česká republika

Abstract

In this paper a condensation of water vapor is solved numerically. Software package OpenFOAM was chosen like a basic tool for solving of problem. Especially for this study the new solver was build. This solver was used for solving of a laminar film condensation on a vertical wall. The result of simulation was compared with analytical solution from Nusselt.

Keywords: steam condensation; numerically solved condensation; laminar film condensation

1. Introduction

Understanding of condensation is very important in many industrial fields e. g. steam condensation behind steam turbine or condensation in a dryer or dishwasher.

This paper helps to understand of steam condensation through numerical solution. Software package OpenFOAM (Open Field Operation And Manipulation) was used for solution of problem. OpenFOAM is a free, open source CFD software that is written in C++ programming language. This software contains a lot of implemented solvers for solving one phase flow, multiphase flow, heat transfer and other problems. Also is possible to use this software for design of new solver for solution particular problem.

The OpenFOAM solver interPhaseChangeFoam [1] was rewritten into new solver interCondensationFoam. Solver interPhaseChangeFoam is for 2-incompressible, isothermal immiscible fluids with cavitation phase change. Uses a volume of fluid phase fraction based interface capturing approach. Governing equations are solved for phases mixture (homogenous equilibrium model). Into this standard solver were implemented energy equation and condensation/evaporation model.

2. Governing equations

Governing equation for conservation of mass, momentum and energy are given by [2], [3] and [4].

Equation for conservation of mass respectively for conversation of volume continuity is shown below (1).

$$\frac{1}{\rho}\frac{\partial p}{\partial \tau} + \nabla \cdot U = (\dot{m}_c + \dot{m}_e)\left(\frac{1}{\rho_l} + \frac{1}{\rho_v}\right) \qquad (1)$$

Conservation of momentum for mixture of two-phases is given by eqn. (2).

$$\frac{\partial \rho\, U}{\partial t} + \frac{\partial \rho\, U}{\partial \tau} + \rho\, U \cdot \nabla U = -\nabla p + \mu\, \Delta U + \rho g + F \quad (2)$$

, where last term on right hand F is for a surface tension in the liquid-gas interface.

The energy equation is given by equation of temperature (3).

$$\frac{\partial T}{\partial t} + U \cdot \nabla T = a\, \Delta T + \left(\frac{\dot{m}_c}{c_{p,l}\, \rho_l} + \frac{\dot{m}_e}{c_{p,v}\, \rho_v}\right) H_v \qquad (3)$$

Continuity phase equation (4) is introduced to track the interface between phases.

$$\frac{\partial \alpha}{\partial t} + \frac{\alpha}{\rho}\frac{\partial p}{\partial \tau} + \frac{\partial \alpha}{\partial \tau} + \nabla \cdot \alpha U = (\dot{m}_c + \dot{m}_v)\frac{1}{\rho_l} \qquad (4)$$

, where α is fluid volume fraction for liquid in this case.

$$\alpha = \frac{V_{phase,cell}}{V_{cell}} \qquad (5)$$

Transport properties in equations (1) - (5) ρ, μ, a, c_p are evaluated for mixture.

The solution is transient and for solving momentum and continuity equation is used PISO algorithm.

3. Condensation/Evaporation model

For condensation/evaporation phenomena in necessary to establish a condensation/evaporation model. Into presented solver was implemented simple model given by [5]. The equations (6) and (7) show this model.

$$-\dot{m}_c = C_c\, \rho_l\, \alpha\, \frac{T - T_{sat}}{T_{sat}}\,,\ for\ T < T_{sat} \qquad (6)$$

$$\dot{m}_v = C_e\, \rho_v\, (1 - \alpha)\, \frac{T - T_{sat}}{T_{sat}}\,,\ for\ T > T_{sat} \qquad (7)$$

, where C_c and C_e are constants that drive condensing respectively evaporating process. The choice of this constants is very important. The temperature at the interface can't go above saturation temperature and vapor temperature must be at saturation temperature.

4. Test case

Solver interCondensationFoam is tested by simulation of laminar film condensation on a wall. This problem is very well known and has approximately solution derived by Nusselt. The approximately solution is described in many books e. g. [6], [7].

* Kontakt na autora: zdenek.sumara@fs.cvut.cz

The case was solved in 2D and computational domain is shown at figure 1.

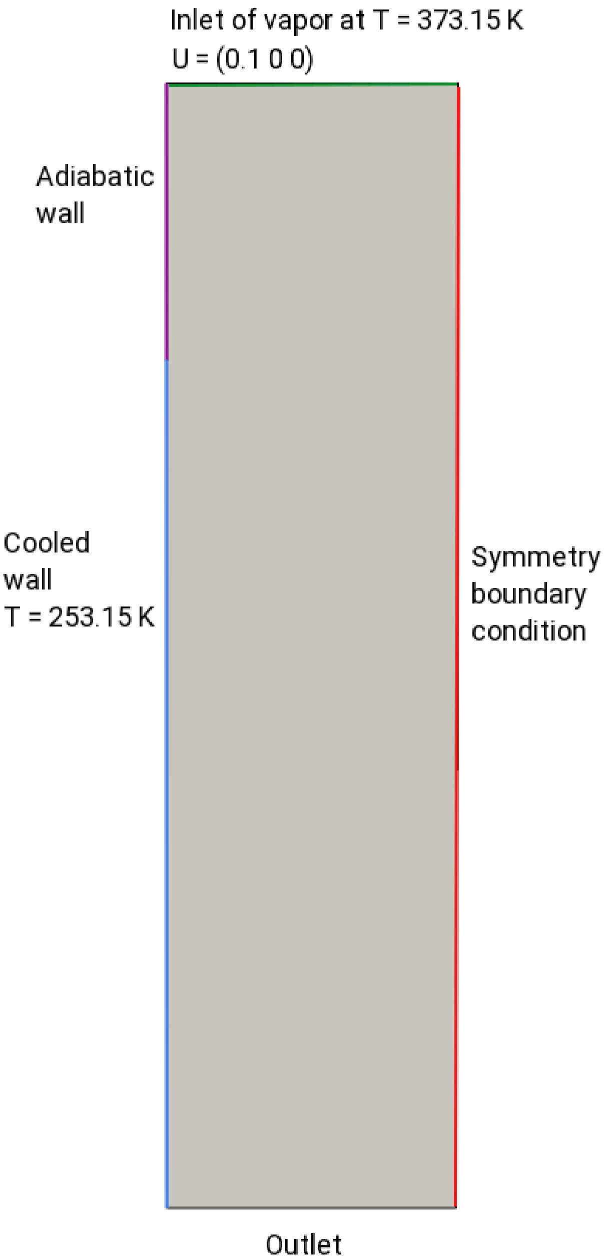

Fig. 1. *Computational domain*

Vapor inflows into domain at low velocity *U = 0.1 m/s* and at temperature *T = 373.15 K* that is equal to saturation temperature. On the right patch is set symmetry boundary condition. First part of left wall is thermally insulated and second part has temperature *T = 353.15 K*. The dimension of domain is *20 x 2.5 (cm)*. 80 thousand hexahedral cells create a mesh. Boundary layer mesh is in the near-wall region.

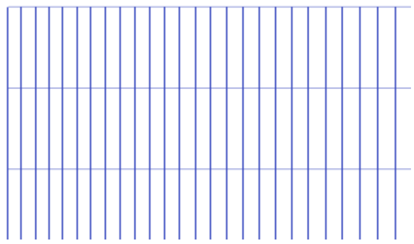

Fig. 2. *Detail of boundary layer mesh*

The height of first cell of boundary layer is *2.5e-5 (m)*.

5. Results

At the figure 4 is shown computation domain with condensate water. The figure 5 shows average heat transfer coefficient calculated from simulation

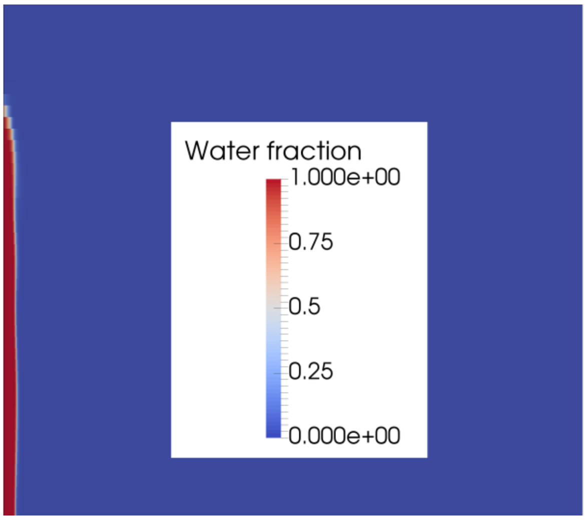

Fig. 3. *Laminar film*

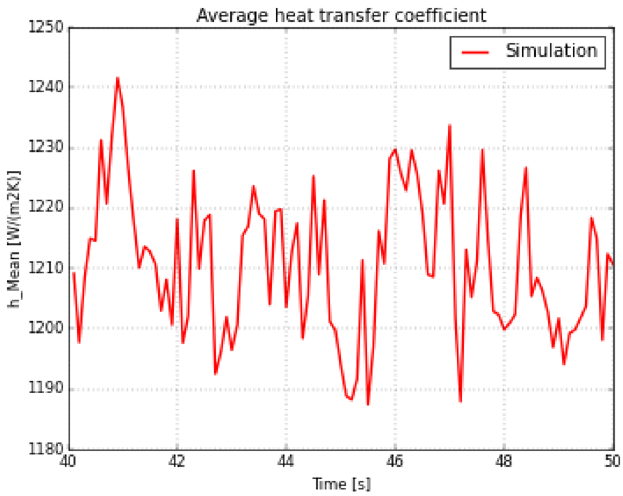

Fig. 4. *Average heat transfer coefficient*

The heat transfer coefficient calculated from simulation is approximately 8 times smaller than coefficient calculated with a help of Nusselt formula.

6. Conclusion

Between results from simulation and from analytical solution is big difference. The reason for this can be in poor condensation/evaporation model. It is also possible that some bugs are in solver interCondensationFoam. It is necessary to check whole code and try to find another model for condensation/evaporation. On the other hand, created solver is working. Evan if it has some bugs, it is definitely first step to numerical solution of steam condensation.

Nomenclature

ρ	density (kg/m^3)
p	pressure (Pa)
τ	pseudo time (s)
U	velocity vector (m/s)
$\dot{m}$	rate of mass transfer (kg/(s m^3))
μ	dynamic viscosity (Pa/s)
g	gravitational acceleration (m/s^2)
F	surface tension force (kg/(m^2 s^2))
T	temperature (K)
a	thermal conductivity (m^2/s)
α	phase fraction (-)
V	volume (m^3)
C	condensation/evaporation model constant

Subscripts

v	vapor
l	liquid
c	condensation
e	evaporation
sat	saturation

Literature

[1] OpenFOAM User Guide. [online], [cit. 17. 4. 2016]. Available from: http://www.openfoam.com

[2] THIELE, R. Modeling of Direct Contact Condensation With OpenFOAM, Stockholm: Royal Institute of Technology, 2010, ISSN 0280-316X

[3] HAIDER, J., Numerical Modeling of Evaporation and Condensation Phenomena, Stuttgart: Universität Stuttgart, 2013

[4] MAMORU, I., TAKASHI, H., Thermo-Fluid Dynamics of Two-Phases Flow, New York: Springer Science + Business Media, Inc., 2006, ISBN-10: 0-387-28321-8

[5] SUN, D., XU, J., CHEN, Q., Modeling of The Evaporation and Condensation Phase-Change Problems with Fluent, Numerical Heat Transfer, Part B, 66: 326-342, 2014, ISSN: 1040-7790

[6] Rohsenow, W. M., Hartnett, J. P., Cho, Y. I., Handbook of heat transfer. 3rd ed., USA: McGraw-Hill, 1998, ISBN: 0-07-053555

[7] Incropera, DeWitt, Bergman, Lavine, Fundamentals of Heat and Mass Transfer. 6th ed., USA: John Wiley & Sons, 2007, ISBN-10: 0-471-45728-0