Nuclear Science and Technology Symposium - NST2016
Helsinki, Finland, 2-3 November 2016

Numerical Modelling of Direct Contact Condensation of Steam in BWR Pressure Suppression Pool System

Giteshkumar Patel, Vesa Tanskanen, Juhani Hyvärinen
LUT School of Energy Systems/Nuclear Engineering,
Lappeenranta University of Technology (LUT), PO Box 20, Fin-53851, Lappeenranta, Finland
giteshkumar.patel@lut.fi, vesa.tanskanen@lut.fi, juhani.hyvarinen@lut.fi

ABSTRACT

This paper presents the results of computational fluid dynamics (CFD) simulations of chugging direct contact condensation (DCC) mode observed in the suppression pool (POOLEX) experiments of Lappeenranta University of Technology. The CFD simulations were done by using Eulerian-Eulerian two-fluid approach of the compressible flow solver of the OpenFOAM CFD code. The interfacial heat transfer between steam and water was modelled by using the surface renewal model of Hughes and Duffey (1991) [1]. Flow turbulence was solved by employing the k-ε turbulence model. Both 2D-axisymmetric and 3D computational domains were modelled and results were compared to the corresponding NEPTUNE_CFD simulations of Tanskanen [2] in order to distinguish possible differences in the solver.

1 INTRODUCTION

In boiling water reactors (BWRs), the safety pressure suppression pool (PSP) systems is one of the key systems during a loss-of-coolant accident (LOCA) or safety valve actuation, which provides a large pressure and heat sink by condensing vapour into liquid and absorbing the energy discharged from a reactor vessel. The prime design purpose of a PSP in BWRs is to mitigate the threats of containment overpressurization and fission product releases. By condensing steam into water and absorbing the energy discharged from a reactor vessel, PSP provides a large pressure and heat sink [3].

During a LOCA or safety relief valve actuation, a large amount of steam and non-condensable gases are blown via the upper drywell of the PSP system to the wetwell compartment through the blowdown pipes. Steam condenses in the water pool and the non-condensables rise up to the gas space of wetwell. As a whole, the discharge of steam into condensation pool is quite complex event which is associated with hydrodynamics and thermodynamics including bubble dynamics, thermal stratification, turbulent mixing, natural circulation, steam condensation within water pool, ducts, and at wall surfaces. Injected steam interacts with pool water by heat transfer, rapid condensation and momentum exchange which induces hydrodynamic loads to the pool structures. Thus, detailed analysis of steam blowdown phenomena either by experiments or with numerical simulations has a great importance from the nuclear reactor safety point of view. In the field of nuclear safety analysis, CFD has become an increasingly applicable tool for thermalhydraulic investigations [4]. However, the published work on CFD simulations of chugging DCC in vertical vent pipes are very limited. The brief review of the numerical efforts conducted by other researchers on condensation pool tests could be found from [5].

The present work contains the CFD simulation of chugging DCC mode appearing in BWR suppression pools. The presented OpenFOAM CFD results are achieved by using the compressible two-phase solver of OpenFOAM which is based on the Eulerian-Eulerian two-fluid approach. The interfacial heat transfer between steam and water is modelled by using the surface renewal theory based condensation model. The experimental references for the analysis and simulations have been obtained from the suppression pool test facility experiments of the Lappeenranta University of Technology. The selected reference case include steam discharge within the POOLEX. Moreover, the OpenFOAM results were discussed together with the corresponding NEPTUNE_CFD simulations of Tanskanen [2] as well.

2 STB-28 POOLEX TEST

A schematic view of the POOLEX test facility is shown in Figure 1. It was a cylinder shaped stainless steel pool with an open top and a conical bottom. The test facility consisted of the steam generator, the steam lines assembly, the blowdown pipe, and the water pool.

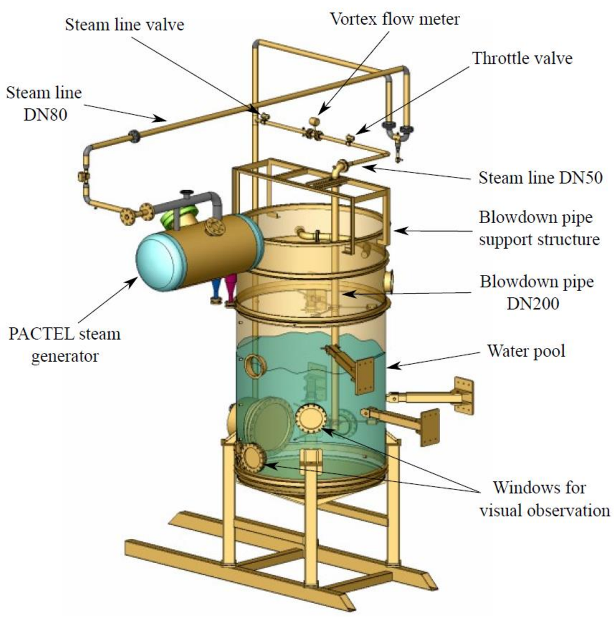

Figure 1: POOLEX test facility.

The inner diameter of the pool was 2.4 m and the height was 5.0 m. A vertical stainless steel blowdown pipe DN200 was placed inside the pool. The blowdown pipe was positioned inside the pool in a nonaxisymmetric location from the vertical axis of pool. The steam generators of the PACTEL facility were used as a steam source during the test. Visual observation of the interior was possible through the circular windows installed in the pool wall. The STB-28 experiment consisted of one long-running steam blowdown (duration 3195 s). The main purpose of this test series was to study the formation and condensation of steam bubbles at the blowdown pipe outlet as a function of pool water temperature. Seven short time intervals of about 12 - 30 s were recorded during the blowdown with a higher sampling rate which were labelled from STB-28-1 to STB-28-7. The temperature of the pool water rose from 47-77 ^{o}C during the test. The steam mass flow rate stayed at the level of 0.3 kg s^{-1} during the whole blowdown.

The sub-test STB-28-4 was selected as the CFD validation case in earlier NEPTUNE_CFD validation studies in the NURESIM and the NURISP EU-projects. Thus it was a suitable choice for the corresponding studies with OpenFOAM. More details about the experimental conditions of the STB-28-4 test can be found from [6].

3 NUMERICAL DETAILS

3.1 Physical models

All results presented in this paper were obtained by using the OpenFOAM CFD solver version 2.3.1. The steam-water system was simulated with the Eulerian two-fluid approach. The transport equations of mass, momentum, and energy for two-phase flow are expressed as

$$\frac{\partial}{\partial t}\left(\alpha_\phi \rho_\phi\right) + \nabla\cdot\left(\alpha_\phi \rho_\phi \mathbf{U}_\phi\right) = \Gamma_\phi, \tag{1}$$

$$\frac{\partial}{\partial t}\left(\alpha_\phi \rho_\phi \mathbf{U}_\phi\right) + \nabla\cdot\left(\alpha_\phi \rho_\phi \mathbf{U}_\phi \mathbf{U}_\phi\right) + \alpha_\phi \nabla\cdot\tau_\phi + \nabla\cdot\left(\alpha_\phi \rho_\phi \mathbf{R}_\phi\right) = -\alpha_\phi \nabla\overline{p} + \alpha_\phi \rho_\phi \mathbf{g} + \mathbf{M}_\phi, \tag{2}$$

$$\frac{\partial}{\partial t}\left(\alpha_\phi \rho_\phi H_\phi\right) + \nabla\cdot\left(\alpha_\phi \rho_\phi \mathbf{U}_\phi H_\phi\right) = \nabla\cdot\left(\alpha_\phi \rho_\phi \mathrm{D}_{\mathrm{T,\,eff},\,\phi} \nabla H_\phi\right) + \Gamma_\phi H_{i,\,\phi} + \mathrm{Q}_\phi, \tag{3}$$

where ϕ indicates an arbitrary phase (afterwards $\phi=a$ for steam and $\phi=b$ for water), α_ϕ is the phase fraction, ρ_ϕ is the density and $\mathbf{U}_\phi$ is the velocity of the ϕ phase. Γ_ϕ is the mass transfer rate. The terms τ_ϕ, $\mathbf{R}_\phi$, $\nabla\overline{p}$, $\mathbf{g}$, and $\mathbf{M}_\phi$ denote the viscous stress tensor, the turbulent stress tensor, the overall pressure gradient, the gravity acceleration and the interfacial momentum transfer between phases, respectively.

The vapour phase heat transfer contribution is negligible compared to the liquid phase one in these simulations with saturated steam. Thus the interfacial heat transfer for liquid phase is defined only. The heat transfer coefficient for the water phase can be defined as

$$\mathrm{h_{i,\,b}} = \frac{\mathrm{Nu_b}\lambda_\mathrm{b}}{\mathrm{L_{t,\,b}}}, \tag{4}$$

where Nu_b, λ_b and $L_{t,\,b}$ indicate the Nusselt number, the thermal conductivity and the characteristic length, respectively. In this study, the condensation model of Hughes and Duffey (HD) [1] has been used and Nu_b is defined as

$$\mathrm{Nu_1} = \frac{2}{\sqrt{\pi}}\mathrm{Re_{t,1}}\mathrm{Pr}^{1/2}, \tag{5}$$

Further details about HD DCC model can be found from [2].

3.2 Computational models

Both 2D-axisymmetric and 3D simulations of the STB-28-4 POOLEX test were conducted. The 2D-axisymmetric grid contains hexahedral cells which was generated by rotating a quadrilateral grid to contain a single cell thickness in respect to z axis (Figure 2). The 2D grid contains 45626 control volumes. The 3D grid contains 302796 hexahedral cells. In order to conform the shape of erupting ellipsoidal bubbles, a spherical curvilinear grid around the blowdown pipe mouth was generated. The grid size around the mouth of the blowdown pipe is approximately 5×5mm.

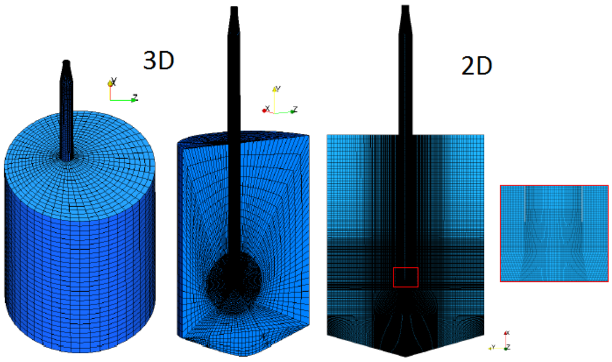

Figure 2: Grids for POOLEX simulations.

The pressure in the liquid volume was initialized by using the hydrostatic pressure. The vapour phase was assumed to be approximately at the saturated state. An inlet velocity of 34.3256ms^{-1} was used which corresponds to the mass flow rate of 0.238445 kgs^{-1} which is the wall condensation corrected value.

4 RESULTS

Figure 3 displays the event of steam jet penetration in the STB-28-4 test. In the beginning, the steam/water interface was inside the blowdown pipe. As steam mass flux increased, the interface moved downwards inside the blowdown pipe and a steam bubble was formed at the pipe outlet. Figure 3 shows the instantaneous contours of steam jet penetration into the water pool in the 3D OpenFOAM simulation, and in the previously simulated 3D NEPTUNE_CFD of [2]. It can be seen that the size of initial bubble in OpenFOAM simulation was larger compared to the experiments and the NEPTUNE_CFD simulations. Also, bubble stayed longer and it travelled further towards the pool bottom. The difference in the results between the CFD codes could be addressed to the differences in compressibility treatment and the interfacial drag modelling. Possible suitability issues of the used grid (cell shapes) for the OpenFOAM solver can be also one reason for errors/differences.

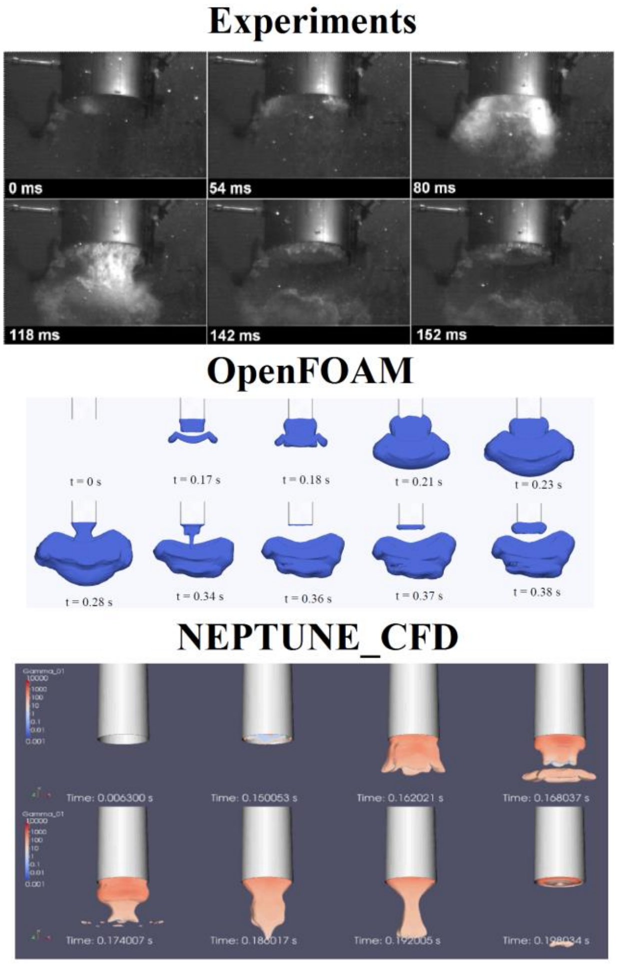

Figure 3: Penetration of the initial steam jet into the water pool.

A longer period of the chugging mode was simulated with a 2D-axisymmetric grid. Figure 4 shows the instantaneous volume fraction fields obtained with OpenFOAM CFD code together with the previously simulated results of NEPTUNE_CFD of [2]. It can be seen that both the codes teach chugging conditions. It seems that the chugging frequency is higher in the OpenFOAM case. The 2D axisymmetric OpenFOAM simulations predicted high enough condensation rates to initiate chugging. Following the collapse of steam bubble, the steam/water interface was retreated inside the blowdown pipe in the upward direction. When the steam pressure was sufficiently high enough to prevent the upward movement of interface, it started to push back the steam/water interface downward again. This chugging cycle continued again from bubble formation to rapid collapse. Figure 5 shows the DCC rate in the 2D-axisymmetric OpenFOAM and NEPTUNE_CFD simulations of the STB-28-4 chugging test. The results indicate that the DCC rate in the 2D-axisymmetric OpenFOAM simulations is relatively high compared to the DCC rate of NEPTUNE_CFD simulations.

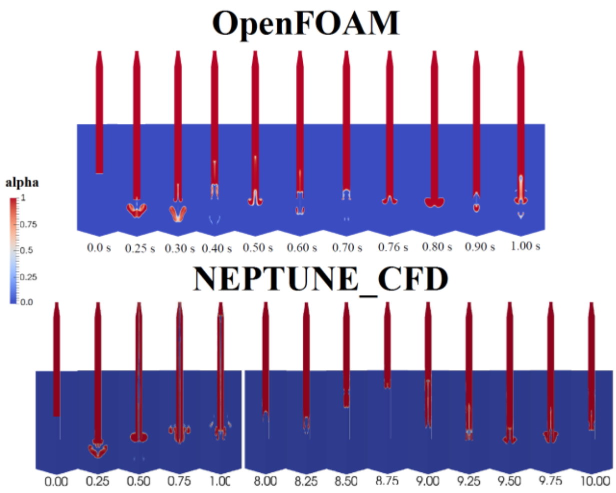

Figure 4: Volume fraction of steam in the 2D-axisymmetric simulations of the STB-28-4 test.

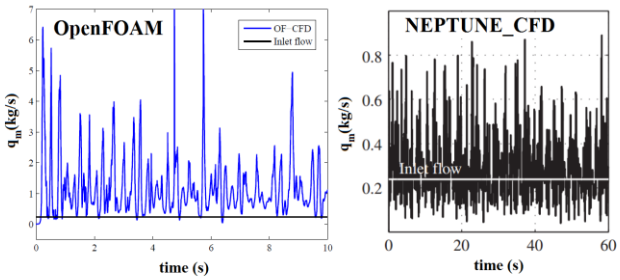

Figure 5: Condensation mass flow rates in the 2D-axisymmetric simulations of the STB-28 test.

5 CONCLUSIONS

The POOLEX STB-28 experiment consisted of one long-running steam blowdown, and the purpose of this test was to analyze the formation and condensation of steam bubbles at the blowdown pipe outlet as a function of pool water temperature by using a high-speed and a standard video camera. Promising numerical simulation results of this test have been obtained previously by [2]. In the present work, the STB-28-4 test case of POOLEX test facility was simulated both with 2D-axisymmetric and 3D computational domains using the Eulerian-Eulerian compressible two-fluid approach of OpenFOAM. The implemented DCC model of Hughes and Duffey was used for interfacial heat transfer calculation. The achieved numerical results were compared with the previously simulated results of NEPTUNE_CFD simulations of [2]. Results showed that the implemented compressible two-phase solver of OpenFOAM is able to invoke chugging phenomenon of open-top condensation pool. However, some qualitative differences (bubble shapes and their sizes) present between OpenFOAM and NEPTUNE_CFD results. The estimated averaged condensation rates and chugging frequency in OpenFOAM simulations are higher compared to the NEPTUNE_CFD results of [2].

ACKNOWLEDGEMENTS

The research leading to these results was funded by the Finnish Nuclear Waste Management Fund (VYR) via SAFIR2014 and SAFIR2018, and YTERA.

REFERENCES

[1] Hughes, E.D., Duffey, R.B., 1991. Direct contact condensation and momentum transfer in turbulent separated flows. Int. J. Multiphase Flow 17, 599–619.

[2] Tanskanen, V., 2012. CFD modelling of direct contact condensation in suppression pools by applying condensation models of separated flow. Acta Universitatis Lappeenrantaensis 472. Lappeenranta University of Technology. ISBN 978-952-265-221-8, ISBN 978-952-265-222-5 (PDF), ISSN 1456-4491.

[3] Lahey, R., Moody, F., 1993. The Thermal-Hydraulics of a Boiling Water Reactor. 2 ed., American Nuclear Society.

[4] Bestion, D., 2012. Applicability of two-phase cfd to nuclear reactor thermalhydraulics and elaboration of best practice guidelines. Nucl. Eng. Des. 253, 311321.

[5] Tanskanen, V., Patel, G., Puustinen, M., Hujala, E., Kyrki-Rajamäki, R., Hyvärinen, J., 2015. CFD modelling of chugging condensation regime of BWR suppression pool experiments, in: The 16th International Topical Meeting on Nuclear Thermal Hydraulics (NURETH-16), Chicago, IL, August 30-September 4, p. 14.

[6] Laine, J., Puustinen, M., 2006. Steam Blowdown Experiments on Chugging. Research Report POOLEX 2/2005. Lappeenranta University of Technology.