Development and validation of CFD methods for nuclear reactor safety assessment (NURESA)

T. Pättikangas[1], V. Hovi[1], I. Karppinen[1], J. Kättö[3], G. Patel[2], J. Peltola[1], T. Rämä[3], V. Tanskanen[2], T. Toppila[3]

[1]VTT Technical Research Centre of Finland Ltd, P.O. Box 1000, FI-02044 Espoo, Finland

[2]Lappeenranta-Lahti University of Technology LUT, P.O. Box 20, FI-53851 Lappeenranta, Finland

[3]Fortum Power and Heat Oy, P.O. Box 100, FI-00048 FORTUM, Finland

Introduction

Computational Fluid Dynamics (CFD) methods have been developed and validated for nuclear reactor safety assessment.

- International benchmark on the stratification erosion of hydrogen layer by a vertical steam jet was participated.
- Thermal stratification of pressure suppression pools have been studied, when steam is injected into pool through SRV spargers and blowdown pipes.
- Models for the Departure from Nucleate Boiling (DNB) have been developed for the OpenFOAM code.
- Co-simulations of NPP components with CFD code and Apros system code have been performed.

Separate effect test of SRV sparger

Sparger experiment performed with the SEF-POOL facility at LUT has been calculated with the OpenFOAM code. Three vapor jets were injected horizontally into a water pool, where chugging condensation occurred.

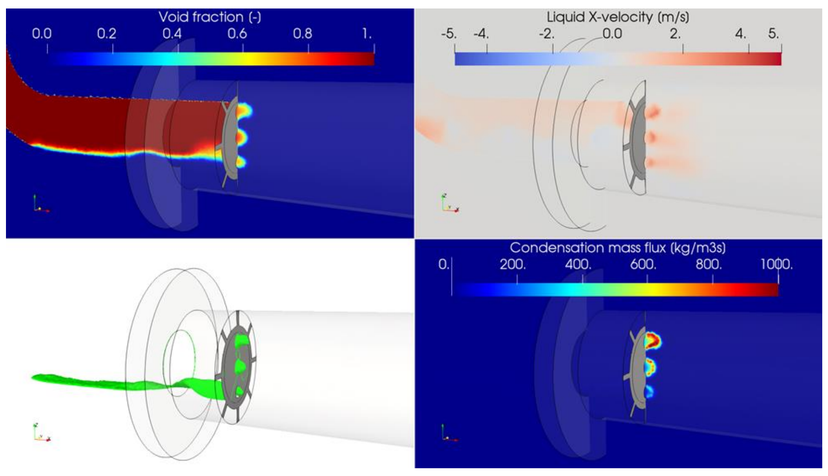

Figure 1. *Chugging condensation of vapor flowing into water pool through a plate with three orifices. Void fraction, iso-surface of void fraction, liquid velocity and condensation mass flux are shown.*

OpenFOAM solver for boiling and DNB

Subcooled nucleate boiling models have been developed and validated for OpenFOAM. The boiling models have been extended to high heat fluxes, where Departure from Nucleate Boiling (DNB) occurs. The models have been included in the public OpenFOAM Foundation release of the solver.

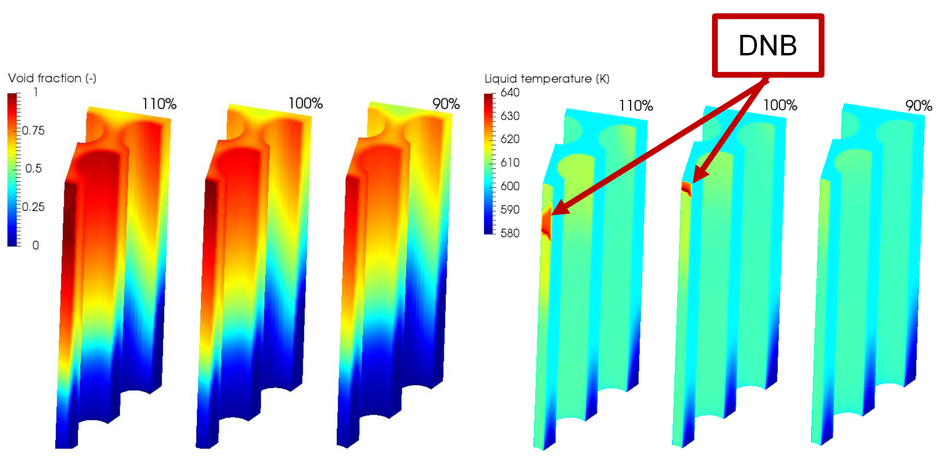

Figure 2. *Void fraction (left) and liquid temperature (right) in a fuel rod bundle of PWR. Results at the power levels, which correspond to 110%, 100% and 90% of the critical heat flux. Rapid increase of temperature is found, when DNB occurs.*

Condensation in pressure suppression pools

Direct contact condensation (DCC) appearing during steam blowdown in BWR pressure suppression pools has been studied. Models for DCC have been developed and validated for the OpenFOAM CFD code. The results have been compared to the POOLEX and PPOOLEX experiments and to the NEPTUNE_CFD code.

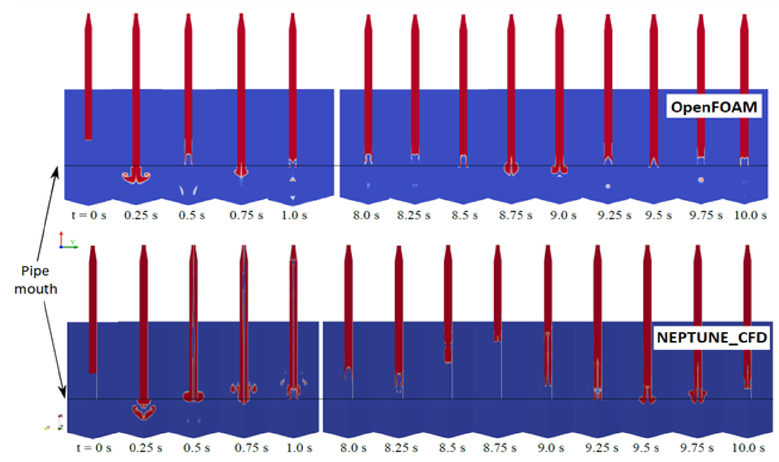

Figure 3. *OpenFOAM and NEPTUNE_CFD simulations of the blowdown of steam into pressure suppression pool. Periodic condensation of steam in the vent pipe is called chugging.*

Coupled CFD-Apros simulations

Coupled CFD-Apros simulations of the pressurizer of generic VVER-440 nuclear power plant have been performed. The behaviour of the pressurizer has been studied during loss-of-feedwater transient.

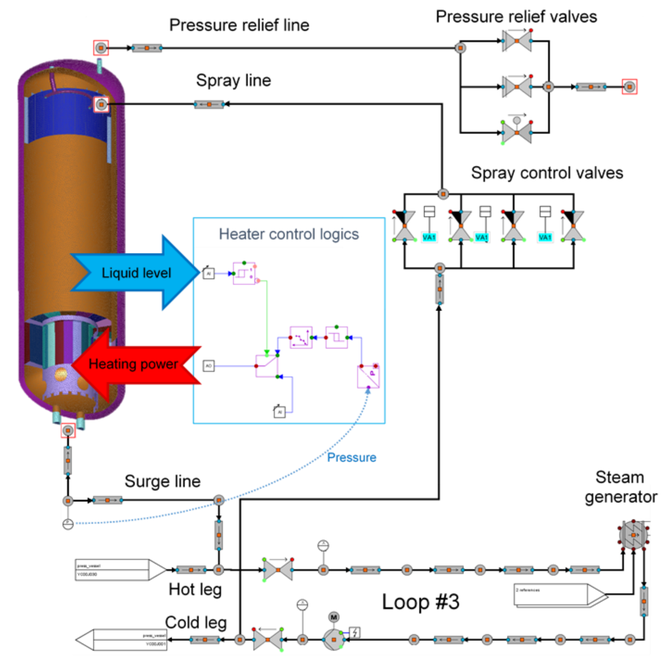

Figure 4. *CFD model of pressurizer was coupled with Apros model of a generic VVER-440 nuclear power plant. The couplings include surge and spray lines, pressure relief line, heating elements and water level measurement.*

Conclusions

- **Computational Fluid Dynamics (CFD) models have been developed and validated in the safety applications, where CFD can be useful.**
- **The commercial ANSYS Fluent and the open-source OpenFOAM codes have been used in the calculations.**

Contact
Timo Pättikangas
Tel. +358 40 595 1968
timo.pattikangas@vtt.fi