Nordic nuclear safety research

NKS-389
ISBN 978-87-7893-475-8

Simulation of PPOOLEX stratification and mixing experiment SPA-T1

Risto Huhtanen, Timo Pättikangas

VTT Technical Research Centre of Finland Ltd

April 2017

Abstract

Thermal stratification of the pressure suppression pool of the PPOOLEX facility has been studied at Lappeenranta University of Technology in experiments, where steam was injected into water pool through a sparger. In the stratification phase of the experiment SPA-T1, steam was injected into the pool at a small mass flow rate of 30 g/s for time 13 650 s. Then the mass flow rate was increased to 123 g/s in order to mix the pool.

In the present report, CFD calculation of the experiment SPA-T1 is presented. The stratification phase and the mixing phase of the experiment were calculated by using the ANSYS Fluent 16.2 CFD code. Single-phase calculation was performed, where the mass, momentum and enthalpy sources of the injected steam were added in front of the sparger holes.

Comparison of the CFD calculation to the measurements shows that the simulation predicts the temperature trends over time rather well. However, during the long stratification phase the calculated mixing between the lower part and the upper part is too strong. This might be corrected by adding grid resolution in the density and velocity gradient layer near the injection. Due to the excessive mixing during the stratification phase the predicted thermal transient in the mixing phase is somewhat milder than in the experiments.

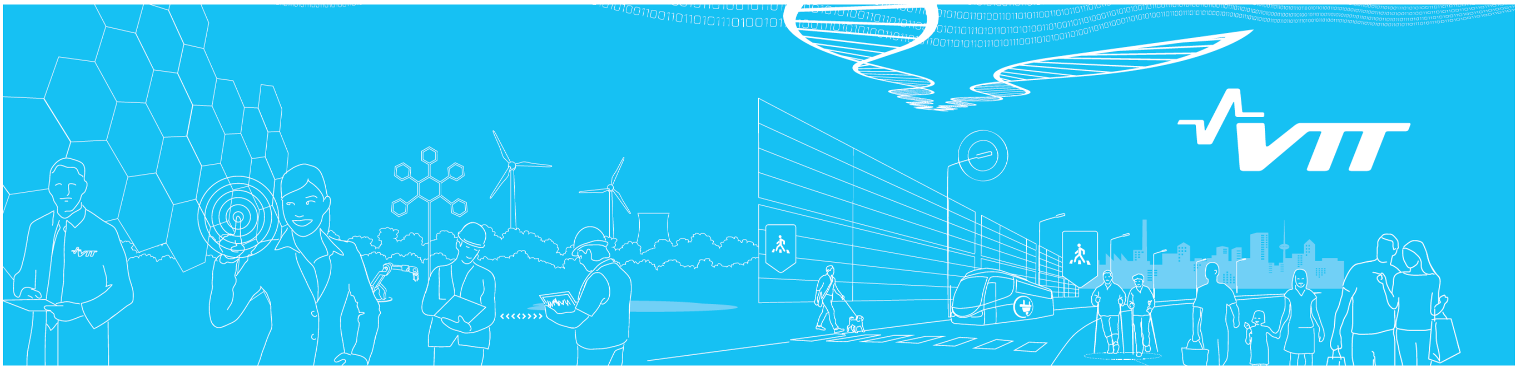
Key words

BWR, pressure suppression pool, condensation pool, stratification, mixing, CFD, computational fluid dynamics

NKS-389
ISBN 978-87-7893-475-8
Electronic report, April 2017
NKS Secretariat
P.O. Box 49
DK - 4000 Roskilde, Denmark
Phone +45 4677 4041
www.nks.org
e-mail nks@nks.org

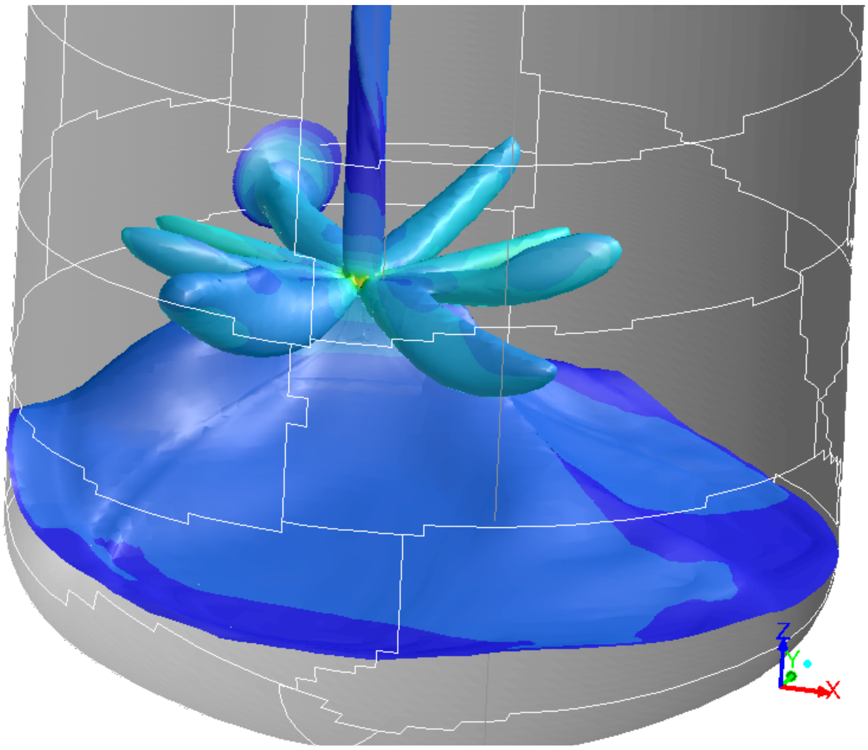
Acknowledgment

NKS conveys its gratitude to all organisations and persons who by means of financial support or contributions in kind have made the work presented in this report possible.



RESEARCH REPORT

VTT-R-01015-17



Simulation of PPOOLEX stratification and mixing experiment SPA-T1

Authors: Risto Huhtanen, Timo Pättikangas

Confidentiality: Public

Report's title	
Simulation of PPOOLEX stratification and mixing experiment SPA-T1	
Customer, contact person, address	**Order reference**
1. Yli-insinööri Jorma Aurela, Valtion ydinjätehuoltorahasto, PL 32, 00023 Valtioneuvosto	SAFIR 14/2016
2. Cristian Linde, Programme Manager NKS-R, Strålsäkerhetsmyndigheten, SE-171 16 Stockholm, Sweden	AFT/NKS-R(16)114/5
Project name	**Project number/Short name**
Development and Validation of CFD Methods for Nuclear Reactor Safety Assessment (NURESA)	108651 / SAFIR_2018_NURESA_2016
Author(s)	**Pages**
Risto Huhtanen, Timo Pättikangas	20
Keywords	**Report identification code**
BWR, pressure suppression pool, condensation pool, stratification, mixing, CFD, computational fluid dynamics	VTT-R-01015-17

Summary

Thermal stratification of the pressure suppression pool of PPOOLEX has been studied at Lappeenranta University of Technology in experiments, where steam was injected into water pool through a sparger. In the stratification phase of the experiment SPA-T1, steam was injected into the pool at a small mass flow rate of 30 g/s for time 13 650 s. Then the mass flow rate was increased to 123 g/s in order to mix the pool.

In the present report, CFD calculation of the experiment SPA-T1 is presented. The stratification phase and the mixing phase of the experiment were calculated by using the ANSYS Fluent 16.2 CFD code. Single-phase calculation was performed, where the mass, momentum and enthalpy sources of the injected steam were added in front of the sparger holes. The turbulence was described with k-ω SST model, where buoyancy terms were added with User Defined Functions.

Comparison of the CFD calculation to the measurements shows that the simulation predicts the temperature trends over time rather well. However, during the long stratification phase the calculated mixing between the lower part and the upper part is too strong. This might be corrected by adding grid resolution in the density and velocity gradient layer near the injection. Due to the excessive mixing during the stratification phase the predicted thermal transient in the mixing phase is milder than in the experiments. Mixing itself is predicted properly.

Confidentiality	Public

Espoo 15.1.2017

Written by	Reviewed by	Accepted by
Risto Huhtanen, Senior Scientist	Timo Pättikangas, Principal Scientist	Lars Kjäldman, Research Team Leader

VTT's contact address
P.O. Box 1000, 02044-VTT, Finland

Distribution (customer and VTT)
Vesa Suolanen, SAFIR2018 Reference Group 4, Programme Manager NKS-R

Preface

This work has been carried out in the Work Package 2 of the NURESA project of the SAFIR2018 programme (The Finnish Research Programme on Nuclear Power Plant Safety). The project has been funded by Valtion ydinjätehuoltorahasto, VTT and NKS (Nordic nuclear safety research). The authors are grateful for comments obtained from the members of the SAFIR2018 Reference Group 4 and from the Northnet Roadmap 3 Reference Group.

Espoo, 15 January 2017

Authors

Contents

1. Introduction

In a Boiling Water Reactor (BWR), the pressure suppression containment typically consists of a drywell and a wetwell with a water pool. The pressure suppression pool serves as a heat sink and steam condenser during postulated accidents and during opening of safety relief valve (SRV). The pressure suppression pool prevents pressure build-up, when steam is vented through blowdown pipes in accidents or through spargers in SRV operation.

In Fukushima Daiichi Unit 3, the pressure build-up in the containment during the first 20 hours after station blackout was attributed to stratification in the pool. Formation and disappearance mechanisms of thermal stratification are therefore under investigation [1]. Addressing stratification and mixing issues in a large pool is thus important and additional data on pool behaviour are needed for the validation of computer models and realistic evaluation of safety margins.

Condensation of steam discharge in the water pool and stratification of heated water has been studied with the PPOOLEX device at Lappeenranta University of Technology [2]. In the experiment, steam is injected to the pool through a sparger. At the interface of steam and water, direct-contact condensation takes place and finally all injected steam is condensed in water. Near the steam discharge, the flow field is two-phase flow, but after condensation only single-phase flow remains, except for a small residue of non-condensable gases. The condensation of steam heats up the pool and may lead to thermal stratification.

Non-condensable gases may be introduced in the steam discharge or they may be released from the liquid phase in the process of steam condensation when dissolved parts of gas fractions are released from the liquid. The non-condensable gases may induce additional mixing and contribute to disappearance of the stratification [3].

Steam flow brings energy to the water pool and the discharge may form density stratification to the pool. This density stratification is stable and may prevent using the whole thermal capacity of the pool to absorb energy. The conditions when this stratification takes place and how it can be broken are examined in this experiment.

In the following, Computational Fluid Dynamics (CFD) simulation of stratification and mixing experiment is performed by using the ANSYS Fluent code. A simplified model is used, where direct-contact condensation is not solved. Instead, it is assumed that the phase change takes place near the discharge hole and the source term of flow is defined so that mass, energy and momentum are conserved.

2. Experimental set-up

The experimental set-up of the PPOOLEX facility is shown in Figure 1. PPOOLEX device is a downscaled model of BWR containment. In the present study, the wet well compartment of the vessel is investigated, where the sparger pipe is located. The inner diameter of the vessel is 2.38 m. It is filled with water up to 3 m height. Above water, there is air volume below the ceiling separating the wet well and dry well compartments. In the CFD model, the water pool is only included, where a rigid surface is assumed. This makes possible to use single-phase CFD modelling. The water surface has a small outlet at the top of the computational domain.

The head of the sparger pipe is shown in Figure 2. Only the lowest row of orifices is open in the experiment SPA-T1. The temperature rake is shown in front of the orifices. A general layout of instrumentation is shown in Figure 3.

In the experiment, there was a short heat-up period in order to remove air from the injection line [2]. During this 200 s long period steam flow rate was about 200 g/s. Temperature of water pool in the experiment SPA-T1 was raised from 13 °C to 16 °C during the injection. This heat-up period was also included in the CFD simulation. However, it was extended to time 250 s because the water pool temperature was not properly heated to the defined value.

The length of the stratification phase after the initial heat-up period was 13 650 s. The steam mass flow during this period was 30 g/s. After stratification phase, there was mixing phase, where mass flow of steam was 123 g/s. The mixing phase lasted for 1 240 s. The experiment continued after this with other stratification and mixing periods, but in this CFD simulation only one cycle was calculated.

Two CFD simulations were performed. In the first simulation, the heat-up period was not used. Instead, the calculation was carefully initialized to correspond to the experimental situation at the end of the heat-up period.

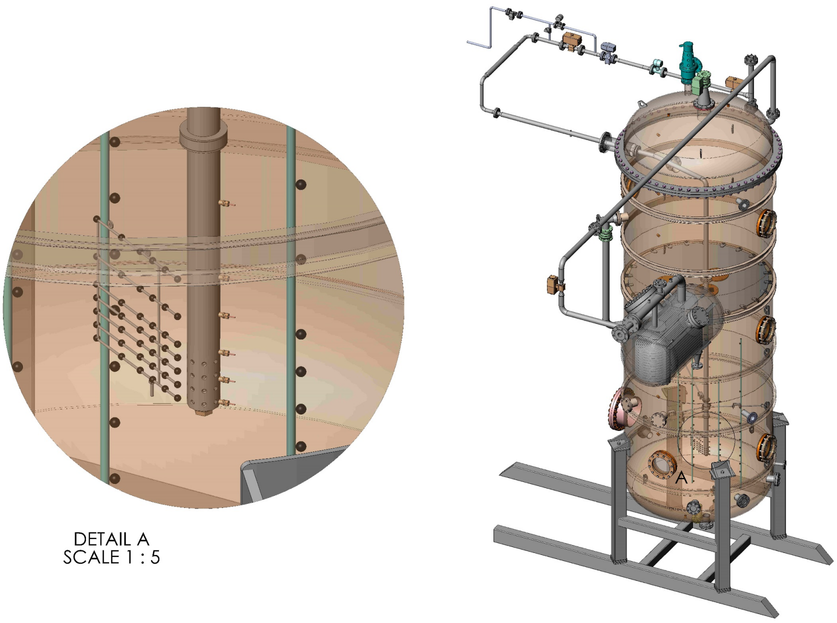


Figure 1. Experimental set-up of the PPOOLEX vessel and detail near the injection pipe [2].

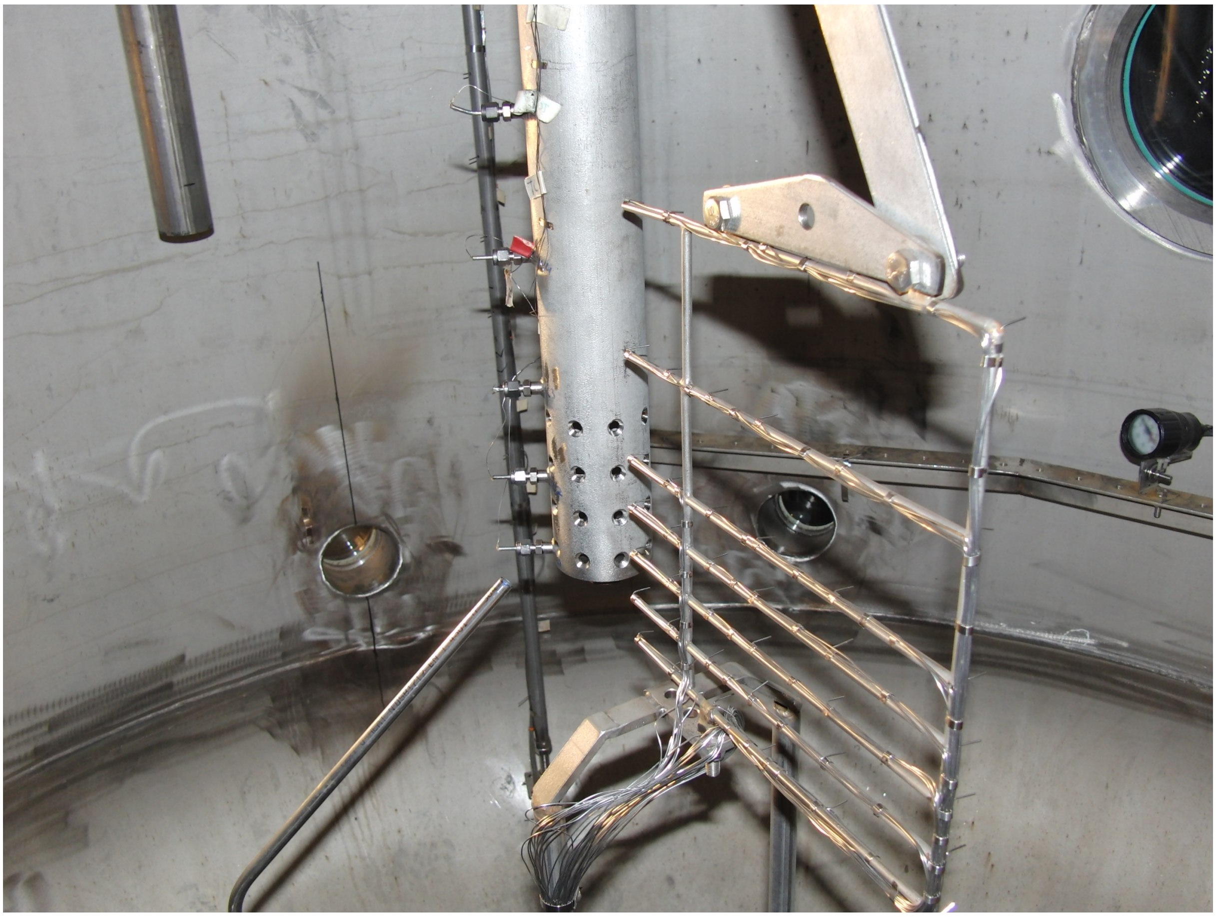
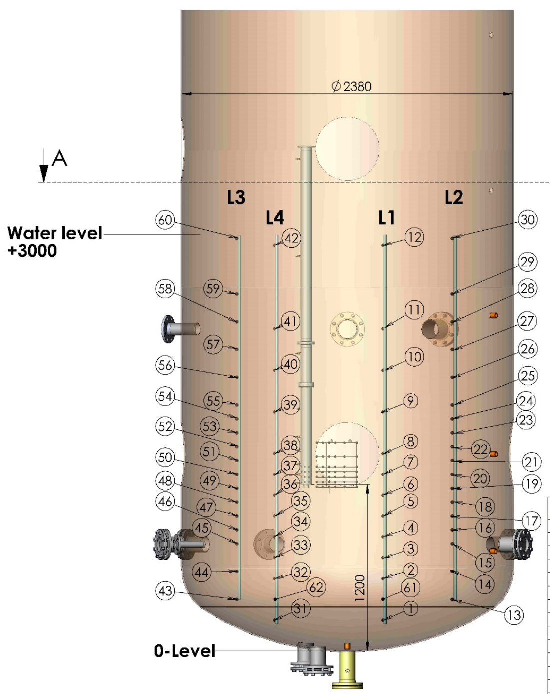
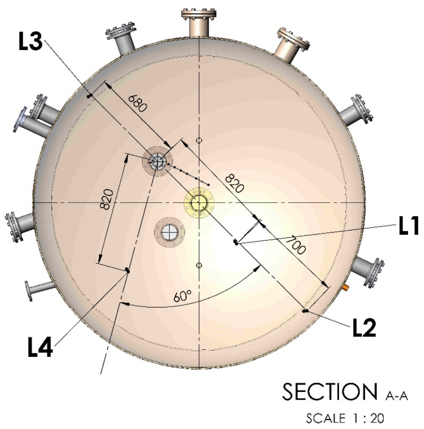


Figure 2. Injection pipe and temperature rake. Only the bottom row of holes is open during the experiment SPA-T1. Picture Markku Puustinen, LUT.





L1		
No.	Code	Height
1	T4100	222
2	T4101	522
3	T4102	672
4	T4103	822
5	T4104	972
6	T4105	1122
7	T4106	1272
8	T4107	1422
9	T4108	1722
10	T4109	2022
11	T4110	2322
12	T4111	2922
61	T4112	372

L2		
No.	Code	Height
13	T4200	372
14	T4201	572
15	T4202	772
16	T4203	872
17	T4204	972
18	T4205	1072
19	T4206	1172
20	T4207	1272
21	T4208	1372
22	T4209	1472
23	T4210	1572
24	T4211	1672
25	T4212	1772
26	T4213	1972
27	T4214	2172
28	T4215	2372
29	T4216	2572
30	T4217	2972

L3		
No.	Code	Height
43	T4300	372
44	T4301	572
45	T4302	772
46	T4303	872
47	T4304	972
48	T4305	1072
49	T4306	1172
50	T4307	1272
51	T4308	1372
52	T4309	1472
53	T4310	1572
54	T4311	1672
55	T4312	1772
56	T4313	1972
57	T4314	2172
58	T4315	2372
59	T4316	2572
60	T4317	2972

L4		
No.	Code	Height
31	T4400	222
32	T4401	522
33	T4402	672
34	T4403	822
35	T4404	972
36	T4405	1122
37	T4406	1272
38	T4407	1422
39	T4408	1722
40	T4409	2022
41	T4410	2322
42	T4411	2922
62	T4412	372

Figure 3. Location of instrumentation in the experiment [2].

3. Numerical model

The flow field was simulated by using ANSYS Fluent version 16.2 [4]. Turbulence was modelled with SST k-ω model with added buoyancy terms. The buoyancy terms are not a standard feature provided by Fluent, but were added into the turbulence model by using User-Defined Functions. The implemented buoyancy terms for turbulence kinetic energy and omega are similar to the ones used in ANSYS CFX [5]. The terms describe the influence of density gradient on turbulence properties. In stable density gradient, they damp turbulence and in unstable case increase turbulence and thus mixing.

The mass flow rate of steam was taken from the experiment SPA-T1 as a function of time. The mass flow rate was equally divided to all 8 orifices. The corresponding mass source of liquid was added to the grid cells near the locations of the orifices. The corresponding momentum and energy sources were added in the same way. The direction of the momentum source was defined to be normal to each orifice.

4. Results of CFD simulations

4.1 On the modelling of turbulence

In the beginning of the first simulation, the standard k-ω turbulence model was applied. With this model, the turbulent viscosity near the injection orifices became very high and the flow field did not have the same form as in the experiment, see Figure 4.

The standard k-ω turbulence model was used only for the beginning of the first run up to 635 s. After that, the SST-version was applied. This led to lower local turbulent viscosity and sharper jets. In the second run, the SST-model was used from the beginning and the results shown later are from this simulation.

4.2 Heat-up period

The initial temperature of water in the heat-up period was 13 °C. Initially the fluid was at rest. During the heat-up period, the whole volume was well mixed due to the high mass flow rate of the injected steam. In the simulation, the length of the injection was extended to 250 seconds because the experimental temperature value of 16-17 °C was not reached within 200 seconds. The velocity and the temperature field at 250 s at the end of heat-up period is shown in Figure 5. The strong jets mix the contents of pool effectively, although there are some small temperature variations in the domain at the end of the transient.

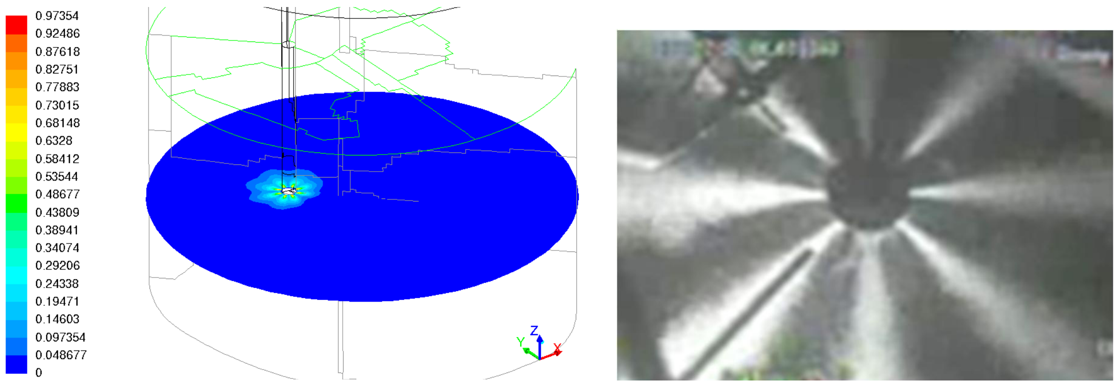


Figure 4. Velocity field at the plane of injection at time 635 s. Photo of the flow field from the experiment shows better penetration of jets (time not specified). Photo is of from Ref. [2].

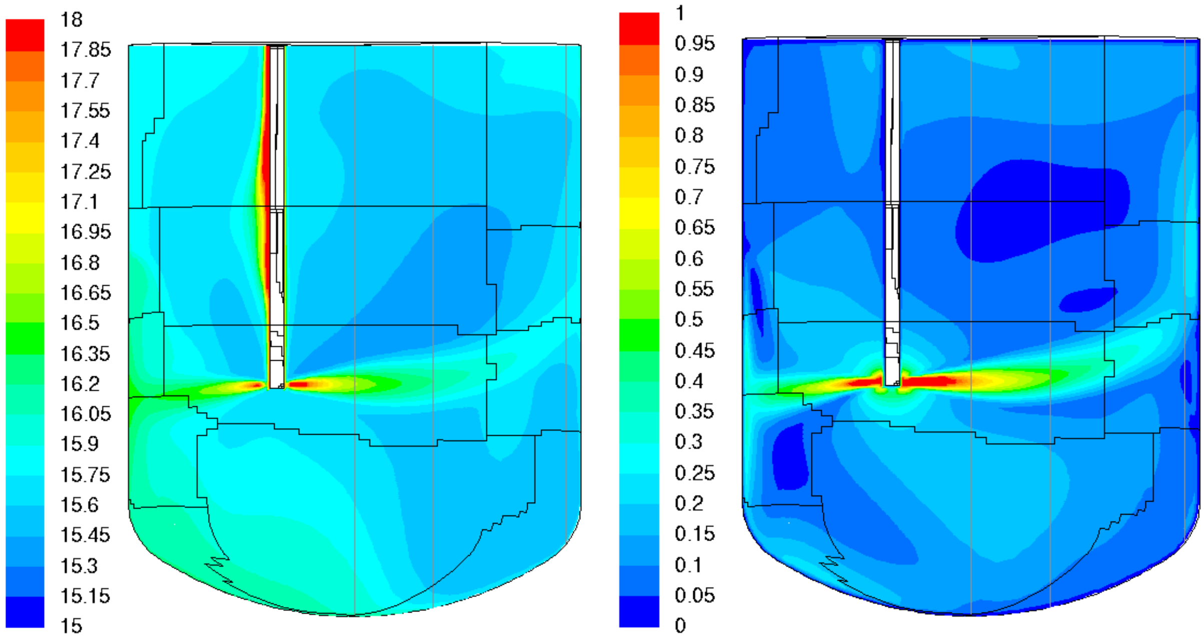


Figure 5. Temperature field (left, in °C) and velocity field (right, in m/s) at 250 s at the end of the heat-up period.

4.3 Stratification phase

In the stratification phase, the injected small mass flow rate of steam gradually heated up the pool above the level of the orifices. The velocity and temperature field at the plane of orifices is shown in Figure 6. The typical liquid velocities in the jets are about 0.2…0.3 m/s.

In Figure 7, the iso-surface of 20 °C is shown coloured by local velocity. The jets of warm water created by condensing steam are clearly visible. In addition, the hot sparger pipe heats up its surroundings and generates an upward plume of warm water.

Development of flow field is shown in Figure 8. The flow field is does not change much during the simulation. The flow velocities of liquid are generally low in the areas other than in the injection orifice. The wall close to the sparger pipe affects the flow, which turns upwards near the wall. Near the bottom of the pool, the flow velocities are very small because of the developing thermal stratification.

Development of temperature field is shown in Figure 9 and Figure 10. During the first 10 000 s, the temperature near the bottom of the pool remains below 20 °C. Above the orifices of the sparger, the water gradually heats up so that the temperature is about 32…33 °C. Thermal stratification is formed, where the temperature gradient is roughly at the distance of 0.5 m from the pool bottom.

Temperature and velocity at the end of the stratification phase at time 13 650 s is shown in Figure 11. The temperature above the injection level is about 38 °C. Near the pool bottom, the temperature is only about 20 °C. Thermal stratification is clear, but it turns out that more mixing takes place in the simulation than the experiment.

In Figure 12, time evolution of the temperature along vertical measurement line L1 is shown (see Figure 3). Gradual formation of temperature gradient at elevation of 0.5 m from the pool bottom is shown. In addition to heating up of water above the spargers, the temperature of the water close to the pool bottom also increases somewhat.

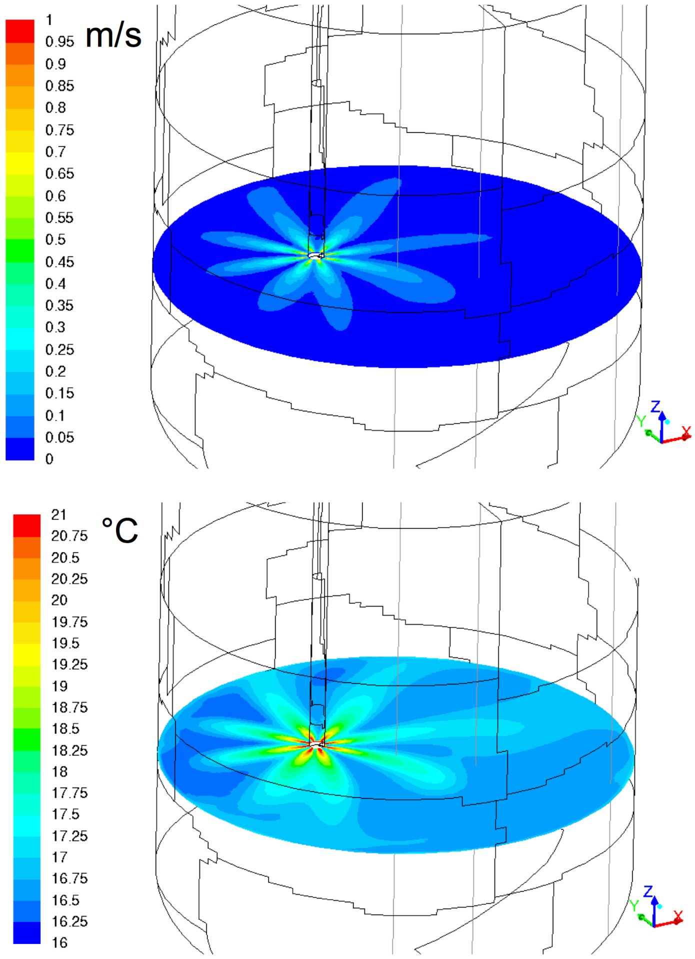


Figure 6. Velocity and temperature field at the plane of orifices. Time is 760 s. The level is z = 1.218 m.

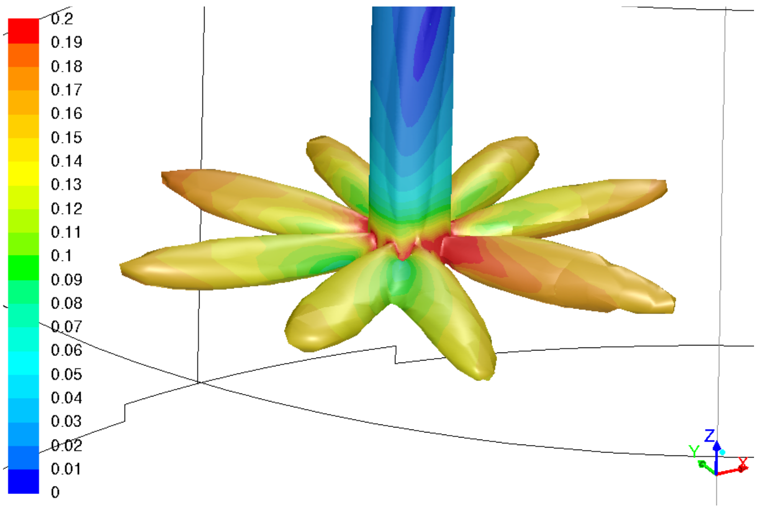


Figure 7. Temperature iso-surface of 18 °C at 76 s. Flow from each orifice can be identified. The hot sparger pipe heats up the liquid on its surface and induces a weaker upward plume around the pipe.

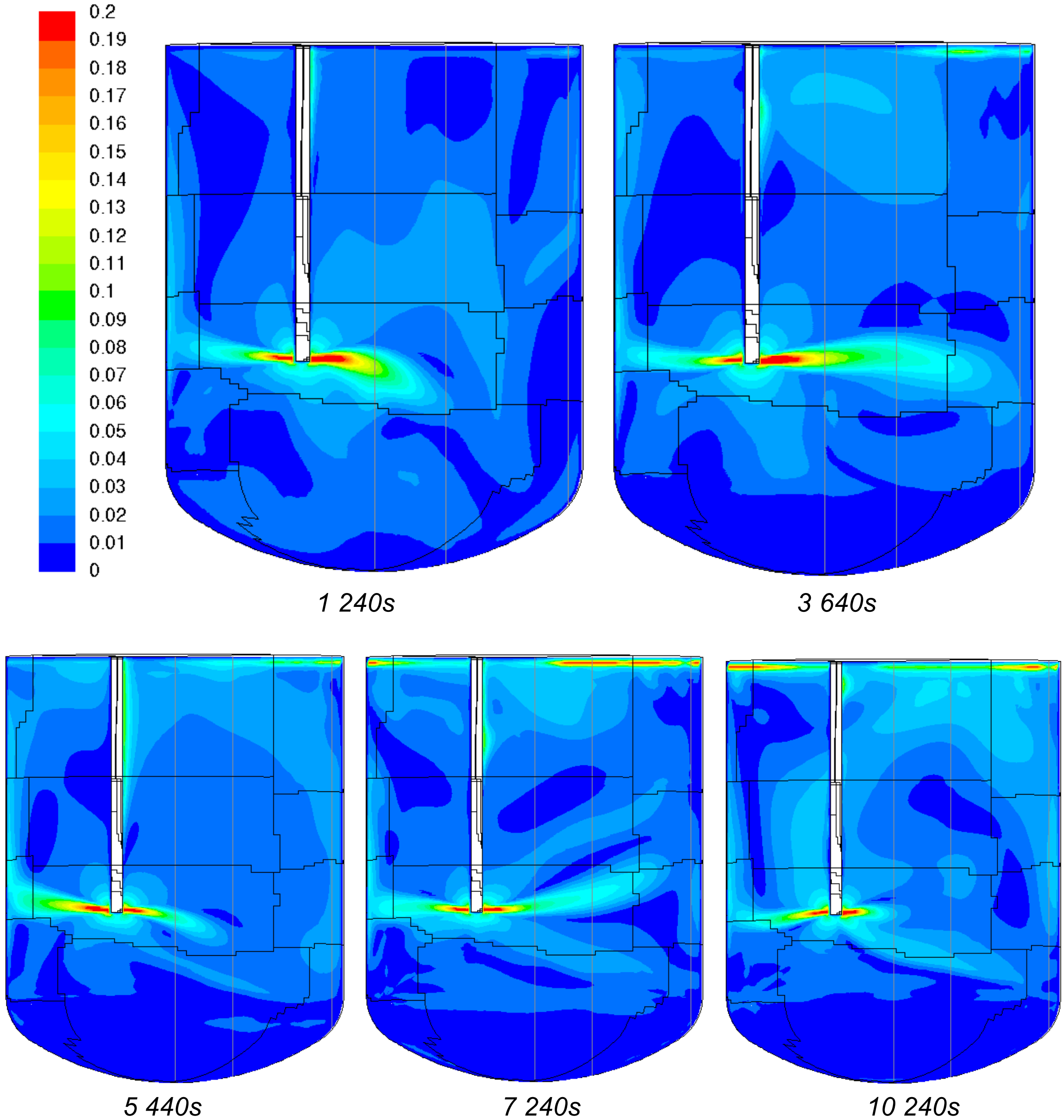
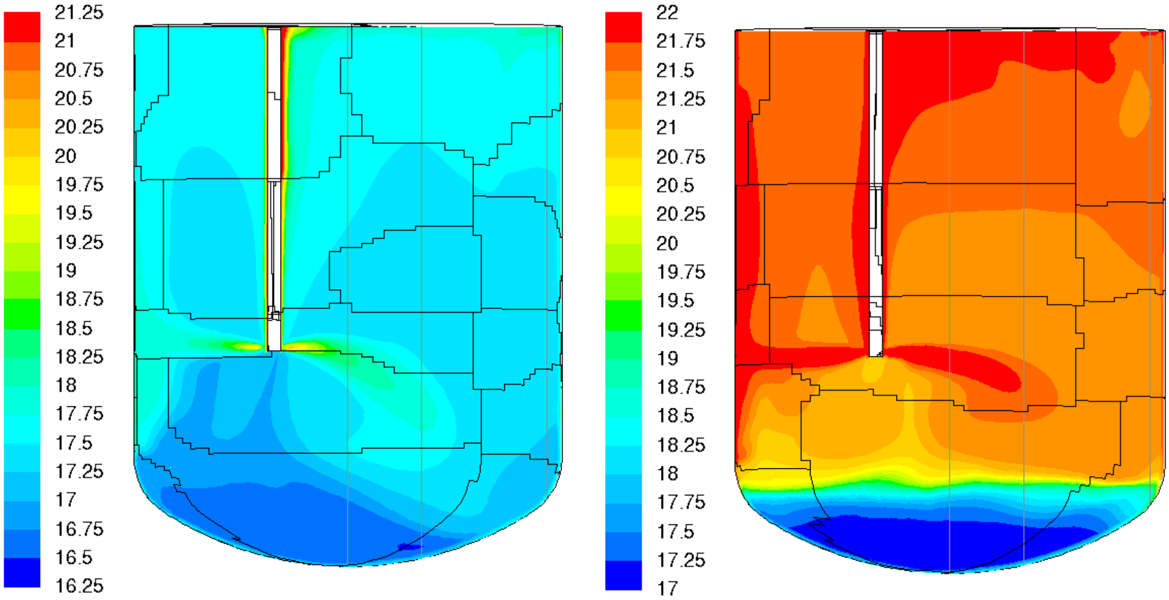
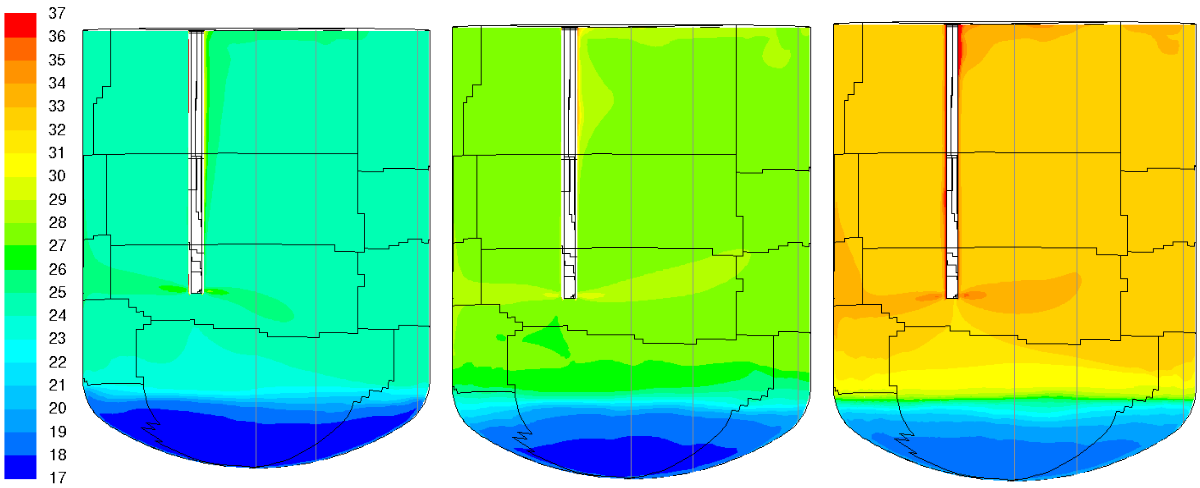


Figure 8. Development of flow velocity (m/s) during the simulation. Flow field keeps mainly its form, but small changes in some plumes are seen. Stratification of cold layer on the bottom of the vessel keeps velocities low there.



1 200s *3 600s*

Figure 9. Development of temperature field in the early phase of the simulation. Scale is in °C.



5 440s *7 240s* *10 240s*

Figure 10. Development of temperature field during simulation. Scale is in °C.

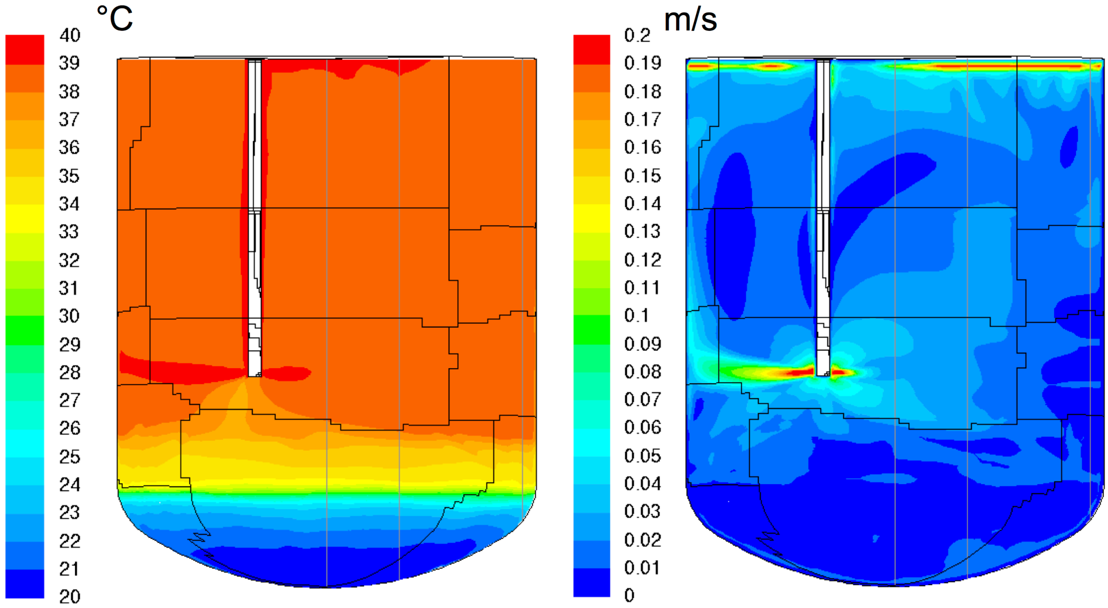


Figure 11. Flow and temperature field after 13 900 s at the end of the stratification phase.

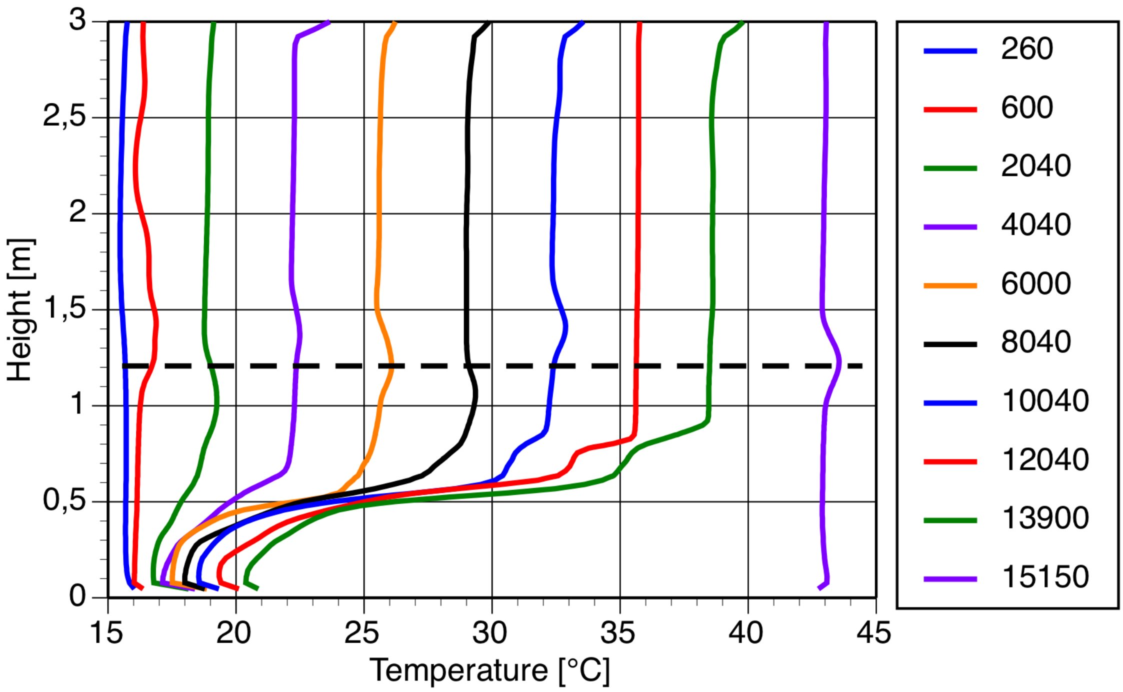


Figure 12. Calculated vertical temperature profile L1 during the stratification phase (0…13 900 s) and after the mixing phase (15 150 s). The dotted line shows the level of injection.

4.4 Mixing phase

In the mixing phase, mass flow rate of steam was increased to 123 g/s in order to break the density stratification in the vessel. Clearly, this was also achieved in the CFD simulation as can be seen in Figure 12. At time 15 150 s, the temperature is almost constant along the vertical measurement line L1. The injected high mass flow rate of steam also increases the average temperature along the measurement line to about 43 °C.

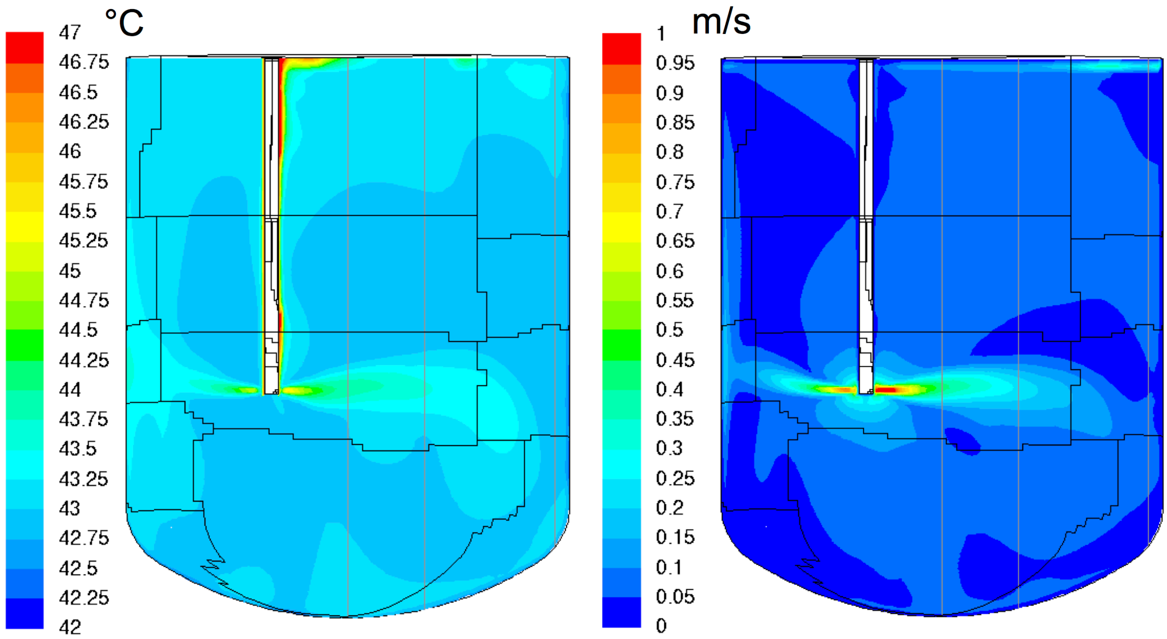


Figure 13. Temperature and flow velocity at time 15 150 s. Temperature differences in the water pool are small.

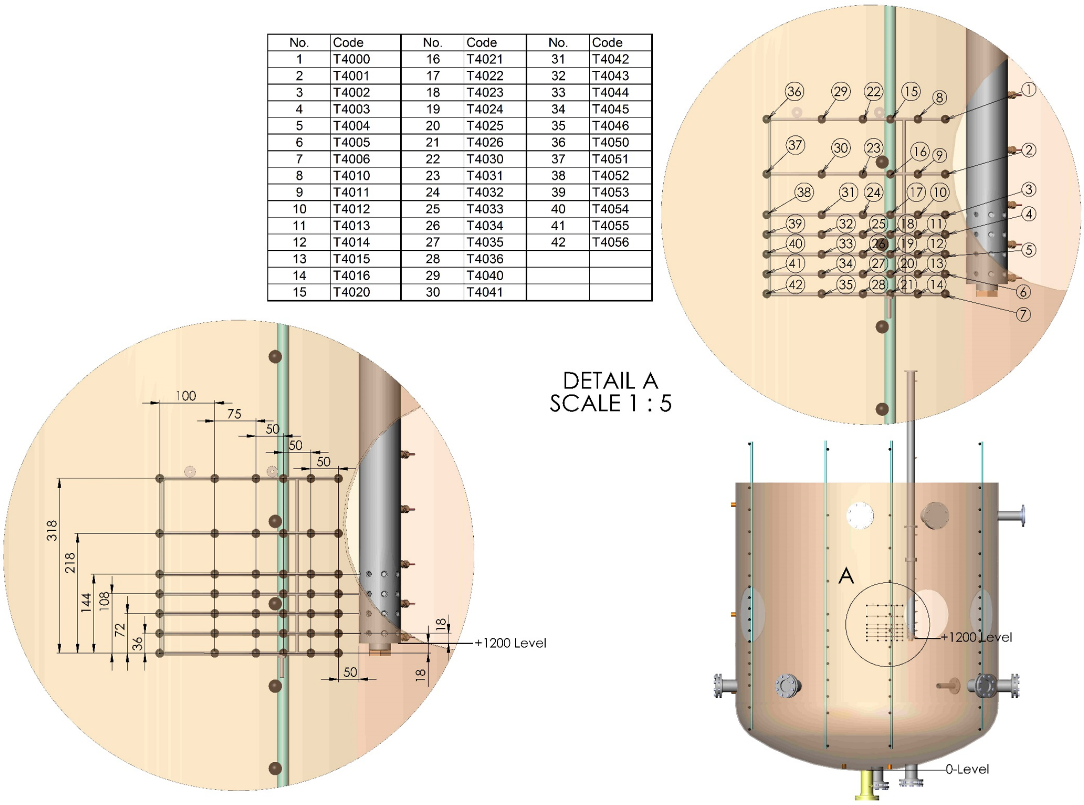
The temperature and flow velocity at the end of the mixing period are shown in Figure 13. The temperature differences in the water pool are very small. Some warm liquid can be seen around the sparger pipe and in the region of the steam jet. The liquid velocities are quite high in the regions of the jets, where flow velocities of 0.5…1 m/s can be seen.

5. Comparison to experimental values

The experiment was instrumented with numerous temperature probes. Comparison between experimental and calculated values are provided with a sample of them. There was a temperature rake near the injection location, shown in Figure 14.

In Figure 15, velocity vectors of liquid water are shown coloured with velocity magnitude in the plane of steam injection. The location of probe T4005 is also shown in relation to the orifices. The probe is located between the jets injected from two orifices.

Measurement point T4106 is located on the measurement line L1 at the elevation of 1.272 m, which is 54 mm above the level of the orifices. Point T4106 can be seen in Figure 3, where it is measurement 7 on the line L1.



No.	Code	No.	Code	No.	Code
1	T4000	16	T4021	31	T4042
2	T4001	17	T4022	32	T4043
3	T4002	18	T4023	33	T4044
4	T4003	19	T4024	34	T4045
5	T4004	20	T4025	35	T4046
6	T4005	21	T4026	36	T4050
7	T4006	22	T4030	37	T4051
8	T4010	23	T4031	38	T4052
9	T4011	24	T4032	39	T4053
10	T4012	25	T4033	40	T4054
11	T4013	26	T4034	41	T4055
12	T4014	27	T4035	42	T4056
13	T4015	28	T4036		
14	T4016	29	T4040		
15	T4020	30	T4041		

Figure 14. Location of measurement points near the injection pipe [2].

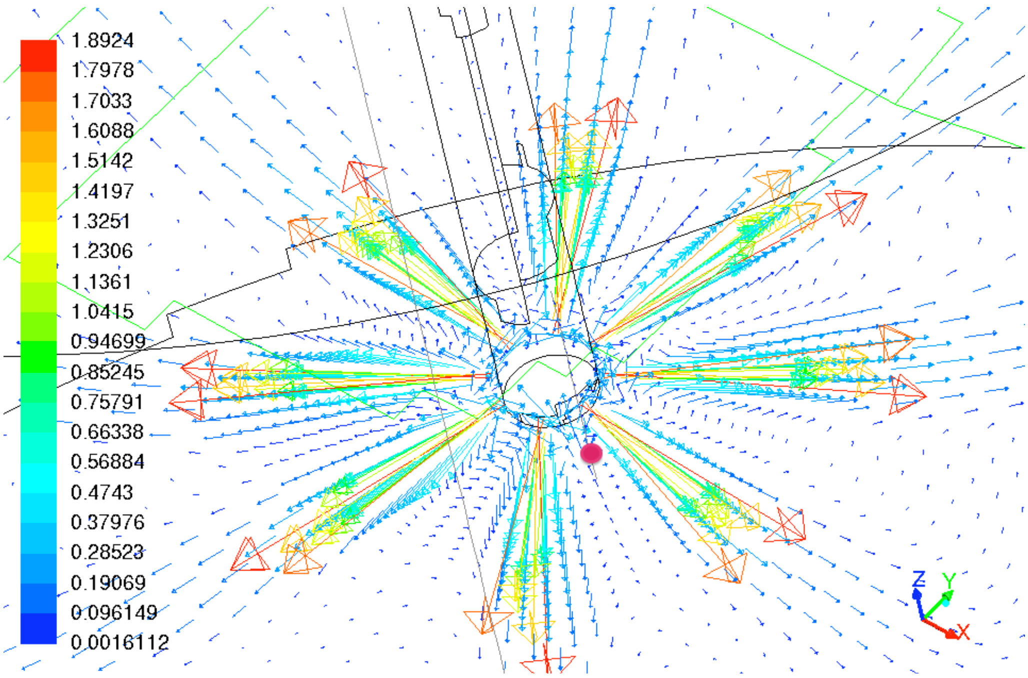


Figure 15. Flow velocity (m/s) near the injection orifices at time 2 735 s. The red dot shows the location of temperature probe T4005 between the jets near the injection plane.

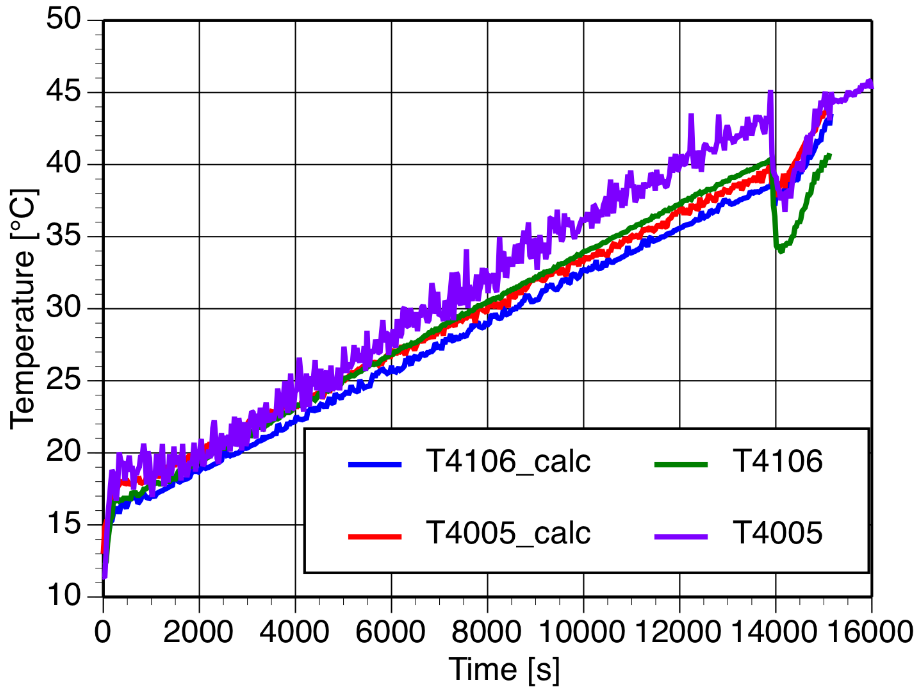


Figure 16. Temperature development at the injection plane, z=1.218 m. Point T4005 in on level z=1.218 m, T4106 in Line 1 at z=1.272 m, see Figure 3 for location.

In Figure 16, the calculated and measured temperature evolutions at points T4005 and T4106 are shown. The temperatures show quite good agreement during the long stratification phase. Difference between the experimental and simulated values is larger at point T4106 further from the injection, where calculated temperatures are somewhat lower than the measured ones. At both points, the temperature transient in the mixing phase is too small in the CFD simulation.

Calculated and measured temperatures on the Line 1 are compared in Figure 17 at two elevations. The point T4109 is clearly above the injection location and the point T4101 should be in the stagnant cold layer. At point T4109, temperature difference between the simulation and experiment grows during time but it is quite small.

At the point T4101, the CFD simulation predicts stronger mixing than is found in the experiment. Even though the cold layer stays rather stagnant in the CFD calculation, there is still too much mixing in the region of the temperature gradient. This might be improved by refining the grid in vertical direction.

In Figure 18, the lowest probe points of Line 1 are shown. The cold layer stays at the bottom of the vessel, but in the simulation it is not as thick as in the experiment. The temperature rise at these locations is very modest.

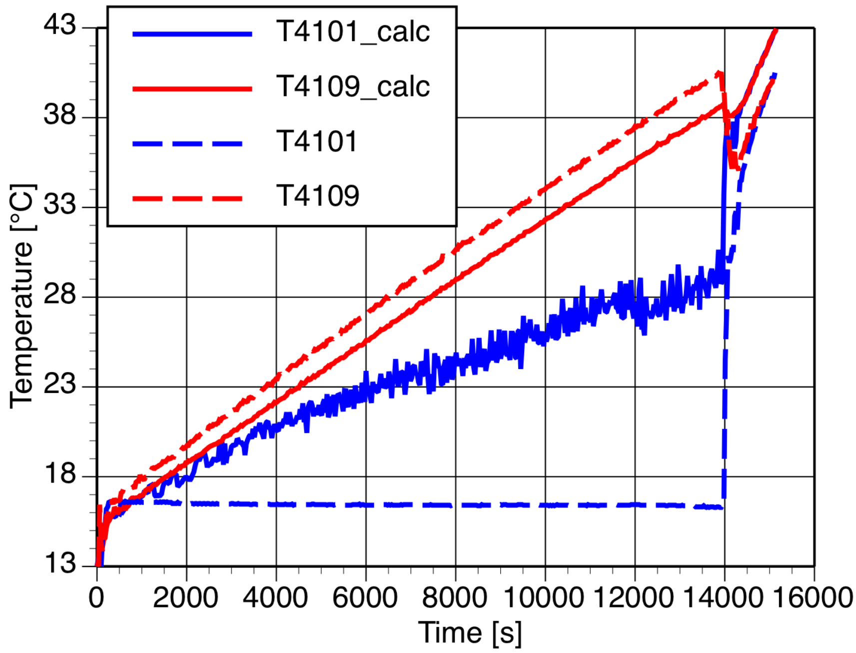


Figure 17. Development of temperature in Line 1 at two locations. T4101 is on elevation 0.522 m, T4109 on elevation 2.022 m.

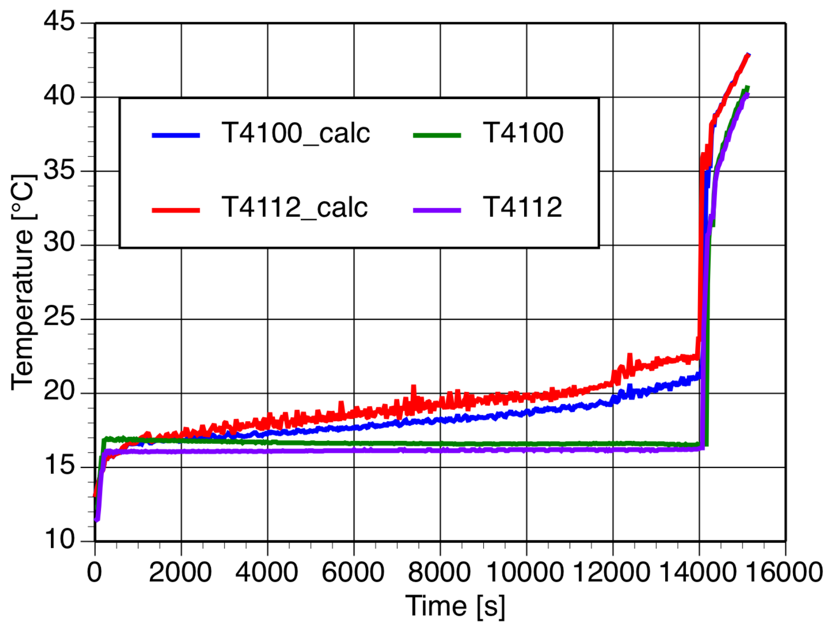


Figure 18. Temperature at the bottom part of vessel. The elevation of T4100 is 0.222 m and T4112 is at the level 0.372 m.

6. Summary and conclusions

The sparger test SPA-T1 performed with the PPOOLEX facility at Lappeenranta University of Technology has been simulated with the ANSYS Fluent CFD code version 16.2. Steam injection from the eight orifices of the sparger was not modelled in geometry. Instead, sources of mass, momentum and energy were defined in the locations of the orifices. The method was found to work properly for this experiment and it generated flow of liquid jets for each injection orifice. The injection was found to be sensitive to the turbulence model used in the simulations and SST k-ω turbulence model was found to work properly.

Density stratification will affect turbulence and this feature was modelled with additional buoyancy terms coded into the SST k-ω model. These terms are not provided in Fluent as a standard feature for k-ω models, but only for k-ε models. Therefore, the buoyancy terms had to be coded with User-Defined Functions into the transport equations of turbulent kinetic energy and omega.

The CFD simulation predicts the temperature trends over time rather well. However, during the long transient the predicted mixing between the cold lower part of the pool and hot upper part is too high. This might be corrected by increasing grid resolution in the density and velocity gradient layer near the injection.

In the experiment, the stratification phase was followed by mixing phase, which was generated by increasing the mass flow rate of steam injection. In the CFD calculation, excessive mixing has occurred already during the stratification phase. Therefore, the calculated thermal transient in the mixing phase was milder than in the experiment. Mixing itself is produced properly.

The heat losses to the environment were taken into account. At the beginning of the experiment, the vessel was colder than the environment and absorbed energy. The heat losses were, however, not very important. The free surface was modelled with an adiabatic wall in the simulation, where a small outlet for water was included. In the experiment, there may be some heat loss due to vaporisation to the air volume above the water layer, which was not included in the simulation.

The single-phase approach that was used in the simulation made possible to use long time steps and perform CFD simulations of stratification and mixing experiment that lasted for more than four hours.

References

1. Byeongnam Jo, Nejdet Erkan, Shinji Takahashi, Daehun Song, Wataru Sagawa, Koji Okamoto, Thermal stratification in a scaled-down suppression pool of the Fukushima Daiichi nuclear power plants. Nuclear Engineering and Design, 305 (2016), pp. 39-50. http://dx.doi.org/10.1016/j.nucengdes.2016.05.017

2. Markku Puustinen, Ari Laine, Antti Räsänen. Additional sparger tests in PPOOLEX with reduced number of injection holes. Research Report INSTAB 1/2015. Lappeenranta University of Technology. Lappeenranta 25.1.2016. 24 p. + app. 15 p.

3. Jiejin Cai, Byeongnam Jo, Nejdet Erkan, Koji Okamoto, Effect of non-condensable gas on thermal stratification and flowpatterns in suppression pool. Nuclear Engineering and Design, 300 (2016), pp. 117-126. http://dx.doi.org/10.1016/j.nucengdes.2016.01.022

4. ANSYS Fluent Theory Guide, Release 16.2, July 2015, section 4.4. ANSYS Inc., PA, USA.

5. ANSYS CFX-Solver Theory Guide, Release 16.2, July 2015, section 2.2.2. ANSYS Inc., PA, USA.

Bibliographic Data Sheet **NKS-389**

Title	Simulation of PPOOLEX stratification and mixing experiment SPA-T1
Author(s)	Risto Huhtanen, Timo Pättikangas
Affiliation(s)	VTT Technical Research Centre of Finland Ltd
ISBN	978-87-7893-475-8
Date	April 2017
Project	NKS-R / COPSAR
No. of pages	20
No. of tables	–
No. of illustrations	18
No. of references	5

Abstract

Thermal stratification of the pressure suppression pool of the PPOOLEX facility has been studied at Lappeenranta University of Technology in experiments, where steam was injected into water pool through a sparger. In the stratification phase of the experiment SPA-T1, steam was injected into the pool at a small mass flow rate of 30 g/s for time 13 650 s. Then the mass flow rate was increased to 123 g/s in order to mix the pool.

In the present report, CFD calculation of the experiment SPA-T1 is presented. The stratification phase and the mixing phase of the experiment were calculated by using the ANSYS Fluent 16.2 CFD code. Single-phase calculation was performed, where the mass, momentum and enthalpy sources of the injected steam were added in front of the sparger holes.

Comparison of the CFD calculation to the measurements shows that the simulation predicts the temperature trends over time rather well. However, during the long stratification phase the calculated mixing between the lower part and the upper part is too strong. This might be corrected by adding grid resolution in the density and velocity gradient layer near the injection. Due to the excessive mixing during the stratification phase the predicted thermal transient in the mixing phase is somewhat milder than in the experiments.

Key words: BWR, pressure suppression pool, condensation pool, stratification, mixing, CFD, computational fluid dynamics

Available on request from the NKS Secretariat, P.O.Box 49, DK-4000 Roskilde, Denmark.
Phone (+45) 4677 4041, e-mail nks@nks.org, www.nks.org