

An Efficient Modeling Method for Thermal Stratification Simulation in a BWR Suppression Pool

NUTHOS-9

Haihua Zhao
Ling Zou
Hongbin Zhang

September 2012

The INL is a
U.S. Department of Energy
National Laboratory
operated by
Battelle Energy Alliance



This is a preprint of a paper intended for publication in a journal or proceedings. Since changes may be made before publication, this preprint should not be cited or reproduced without permission of the author. This document was prepared as an account of work sponsored by an agency of the United States Government. Neither the United States Government nor any agency thereof, or any of their employees, makes any warranty, expressed or implied, or assumes any legal liability or responsibility for any third party's use, or the results of such use, of any information, apparatus, product or process disclosed in this report, or represents that its use by such third party would not infringe privately owned rights. The views expressed in this paper are not necessarily those of the United States Government or the sponsoring agency.

An Efficient Modeling Method for Thermal Stratification Simulation in A BWR Suppression Pool

Haihua Zhao*, Ling Zou, Hongbin Zhang

Idaho National Laboratory

* Corresponding author: PO BOX 1625, Idaho Falls, ID 83415-3870, USA

Haihua.Zhao@inl.gov, Ling.Zou@inl.gov, Hongbin.Zhang@inl.gov

ABSTRACT

The suppression pool in a boiling water reactor (BWR) plant not only is the major heat sink within the containment system, but also provides the major emergency cooling water for the reactor core. In several accident scenarios, such as a loss-of-coolant accident and extended station blackout, thermal stratification tends to form in the pool after the initial rapid venting stage. Accurately predicting the pool stratification phenomenon is important because it affects the peak containment pressure; the pool temperature distribution also affects the NPSHa (available net positive suction head) and therefore the performance of the Emergency Core Cooling System pumps that draw cooling water back to the core. Current safety analysis codes use zero dimensional (0-D) lumped parameter models to calculate the energy and mass balance in the pool; therefore, they have large uncertainties in the prediction of scenarios in which stratification and mixing are important. While three-dimensional (3-D) computational fluid dynamics (CFD) methods can be used to analyze realistic 3-D configurations, these methods normally require very fine grid resolution to resolve thin substructures such as jets and wall boundaries, resulting in a long simulation time. For mixing in stably stratified large enclosures, the BMIX++ code (Berkeley mechanistic MIXing code in C++) has been developed to implement a highly efficient analysis method for stratification where the ambient fluid volume is represented by one-dimensional (1-D) transient partial differential equations and substructures (such as free or wall jets) are modeled with 1-D integral models. This allows very large reductions in computational effort compared to multi-dimensional CFD modeling. One heat-up experiment performed at the Finland POOLEX facility, which was designed to study phenomena relevant to Nordic design BWR suppression pool including thermal stratification and mixing, is used for validation. The results from the GOTHIC lumped parameter based analysis are used to obtain boundary conditions for BMIX++ code and CFD simulations. Comparisons between the BMIX++, GOTHIC, and CFD calculations against the POOLEX experimental data are discussed in detail.

KEYWORDS

Suppression pool, thermal stratification, simulation, and CFD

1. INTRODUCTION

The suppression pool in a BWR plant not only is the major heat sink within the containment system, but also provides the major emergency cooling water for the reactor core because it stores the largest amount of water within the containment. In passive safe BWRs, such as General Electric's ESBWR design, the long-term post-accident containment pressure is determined by a combination of the noncondensable gas pressure and vapor partial pressure in the wet well gas space. The suppression pool surface temperature, which determines the vapor

partial pressure, is very important to the overall containment pressure response [1]. Separate-effects tests with a one-tenth scaled-down ESBWR suppression pool at Purdue University indicated that significant thermal stratification is likely to exist after the steam blowdown and direct contact condensation in the pool [2]. At this stage, the air fraction in the vent flow becomes lower and the pool temperature becomes higher. For the current operating BWR plants, thermal stratification also tends to form in the pool after the initial rapid venting stage during LOCA (loss of coolant accident) transients. This phenomenon has been studied experimentally. For example, the POOLEX experiments performed at Lappeenranta University of Technology, Finland were designed to study phenomena relevant to the Nordic design BWR suppression pool, including thermal stratification and mixing [3]. Two heat-up tests demonstrated strong thermal stratification up to 37°C within the water pool. Similar phenomena exist during extended station blackout accidents (SBO) because no active power is available to break down the thermal stratification in the suppression pool. Without taking into account the thermal stratification, any analysis predicting the containment pressure response would have large uncertainties. Therefore, the thermal stratification of the suppression pool is of primary importance for the safety analysis of those accidents.

The suppression pool temperature distribution also affects the NPSHa and, consequently, the performance of the Emergency Core Cooling System pumps, which draw cooling water from the suppression pool back to the reactor core. NPSHa can be calculated by:

$$NPSH_a = \left(P_{nc} + P_v(T_{surface}) - P_v(T_{suction}) \right) / \rho g + \Delta H - H_{loss} \quad (1)$$

where P_{nc} is the noncondensable gas pressure, $P_v(T_{surface})$ the vapor pressure determined by the pool surface temperature, $P_v(T_{suction})$ the vapor pressure determined by the water temperature at the suction point that typically is located at the lower part of the pool, ΔH the elevation difference between the water surface and the suction point, and H_{loss} the pressure loss on the suction side converted to head. If the water pool is treated as one single volume, $T_{surface} = T_{suction}$, then Eq. 1 reduces to

$$NPSH_a = P_{nc} / \rho g + \Delta H - H_{loss} \quad (2)$$

If strong thermal stratification in the suppression pool exists, the surface temperature could be much higher than the suction point temperature, which results in $P_v(T_{surface}) - P_v(T_{suction}) > 0$. This is a potential large margin for NPSHa. For example, if the pool surface temperature is 70°C with 1 atm pressure and the suction point temperature is still in the initial value 30°C, the resulting increased head for NPSHa is $(0.31161 \text{ bar} - 0.042417 \text{ bar}) / (9.8 \text{ m/s}^2) \times 0.001 \text{ m}^3/\text{kg} = 2.75 \text{ m}$. For a typical NPSHa value of 9 m [4], this is about $2.75 \text{ m} / 9 \text{ m} = 30\%$ of margin increase. This effect had not been analyzed in the existing NPSH sensitivity analysis [4], though this effect is far larger than any single effect considered in the analysis, such as 5% uncertainty for decay heat power and 2% for initial pool temperature or initial wet well pressure. Therefore, detailed thermal stratification analysis should be included in the long-term BWR safety analysis in order to accurately simulate the transients and recover this potential large safety margin.

Besides the suppression pool, mixing and stratification phenomena play major roles in the safety of reactor systems with large containments and enclosures. For both current operating light water reactors with active safety features and advanced light water reactors with passive safety features, post-LOCA gas transport between containment compartments and hydrogen

distribution have been identified by several international expert groups as high-ranking phenomena because they mostly affect the risk of containment failure [5]. Buoyancy-driven flows, potentially augmented by break-jet momentum, play a key role in gas transport [6]. More generally, enclosure flows driven by free buoyant jets and wall boundary layers are important in nuclear systems experiencing fires, using passive autocatalytic recombiners, and removing aerosols following severe accidents. Several large test facilities, such as Swiss PANDA [5] and German THAI [7], have been used to continuously investigate these phenomena over the past decade. Long-term passive containment cooling in AP-1000 design is another example where mixing and stratification phenomena are important. AP-1000 design uses a passive containment cooling system to remove decay heat. Mass transfer is the dominant means of containment heat removal on both inner and outer steel shell surfaces [8]. On the inside, condensation on the containment shell dominates heat removal and is strongly influenced by distribution of steam and noncondensable gases. The containment design used several highly conservative assumptions regarding mixing and condensation. Improved thermal mixing modeling capability would increase the confidence on the passive containment performance and potentially allow further power uprate to improve plant economics.

In terms of modeling and simulation efforts in large enclosure mixing, two opposite trends can be observed [9]. One is along the traditional system analysis approach by using decoupled, highly simplified and conservative 0-D models to study mixing in large enclosures. Another path is to try expensive and inefficient 3-D CFD simulations. Current major system safety analysis or severe accident analysis codes (such as RELAP5 [10], TRAC [11], MELCOR [12], etc.) either have no models or only 0-D models for thermal mixing and stratification in large enclosures. The SASSYS code developed by Argonne National Laboratory, one of the major system analysis codes for sodium-cooled fast reactors, only provides lumped-volume-based 0-D models that can only give very approximate results and can only handle simple cases with one mixing source [13].

Two-dimensional (2-D) or 3-D CFD codes have been widely used to simulate thermal stratification and mixing for laminar problems and some turbulent problems with simple geometric configurations. Krepper et al. used CFX-4 to investigate natural convection and thermal stratification in a water pool heated by a horizontally emergency condenser [14]. The calculated pool surface temperature error is around 20°C for a total 50°C measured temperature difference within the pool after 2000 s heat-up time. Gupta et al. studied thermal stratification in a side-heated water pool for advanced heavy water reactor applications with the Lam–Bremhorst κ – ϵ turbulence model [15]. The calculated pool surface temperature error is around 1.5°C for a total 5.5°C measured temperature difference within the pool after 1500 s heat-up time. GOTHIC [16], a containment analysis code developed by EPRI (the Electric Power Research Institute), has both a lumped parameter-based method and CFD-like field simulation capability. Li and Kudinov [17] used GOTHIC code to simulate a 2-D mixing process in the water pool heat-up experiment performed at POOLEX facility. The spatially converged result shows 4°C over prediction for the water surface temperature, compared to a 37°C temperature difference within the pool at the end of 14000 s heat-up process. To address the difficulty of using GOTHIC to simulate oscillatory motion of the water in the blowdown pipes caused by direct contact condensation phenomena, Li et al., proposed to model the effect of steam injection on the mixing and stratification with the effective heat source model and the effective momentum source model [18].

In addition to the suppression pool analysis, containment mixing and gas transport

phenomena also have been simulated with 2-D or 3-D CFD methods by many researchers [5, 7]. These simulations typically take a very long time to run. In the framework of the fifth EU-FWP project ECORA, the CFD capabilities for simulating flows in the containment of nuclear reactors were evaluated [19]. The assessment included a first attempt to use best practice guidelines for the analysis of long, large-scale, transient problems. Because of the large computational overhead of the analysis, it was concluded that the application of the best practice guidelines to full containment analysis is out of reach with the currently available computer power. Without fully following best practice guidelines, the CFD simulation uncertainties cannot be quantified.

Considering the limitations of the inadequate 0-D methods and the inefficient 3-D CFD methods, new accurate and efficient thermal mixing and stratification methods are needed to improve analysis accuracy and reduce modeling uncertainties, especially for system safety analysis. A middle path exists that would provide adequate physical insights with combinations of different 1-D methods. Previous scaling analysis [6] has shown that stratified mixing processes in large stably stratified enclosures can be described using 1-D partial differential equations, with the vertical transport by free and wall jets modeled using standard integral techniques, which can have different varying flow directions besides the vertical direction. This allows very large reductions in computational efforts compared to 3-D numerical modeling of turbulent mixing in large enclosures. The BMIX++ code was originally developed at University of California at Berkeley to implement such ideas [20, 21, and 22]. The next section gives a brief overview on BMIX++ code and its modeling methods.

2. BMIX++ CODE MODELING METHODS

Depending on mixing sources strength and the aspect ratio, scaling analysis [6] has shown that the ambient fluid between jets and boundary layer flows tends to organize into either a homogeneously mixed condition or a vertically stratified condition that can be described by a 1-D temperature and concentration distribution. Thus, stratified mixing processes in large complex enclosures can be described by using 1-D differential equations, with transport in free and wall jets modeled using 1-D integral models. The detailed geometry of the enclosure becomes unimportant, and only the horizontal cross-sectional area and perimeter must be specified as a function of elevation. For the stratified enclosure, the governing equations for the ambient fluid can be derived and written in the following compact form:

$$A(z) \frac{\partial \mathbf{G}}{\partial t} + \frac{\partial \mathbf{F}}{\partial z} = \mathbf{S}, \quad (3)$$

where $A(z)$ is the horizontal cross-sectional area of the volume at elevation z , and $\mathbf{G}$, $\mathbf{F}$, and $\mathbf{S}$ are the vectors of conserved quantities, fluxes, and source terms, respectively,

$$\mathbf{G} = \begin{pmatrix} \rho \\ 0 \\ \rho h \\ \rho \chi_1 \\ \vdots \\ \rho \chi_{ns-1} \end{pmatrix} \quad \mathbf{F} = \begin{pmatrix} \rho Q_{sf} \\ P \\ \rho h Q_{sf} - Ak \frac{\partial T_{sf}}{\partial z} \\ \rho \chi_1 Q_{sf} - \rho AD \frac{\partial \chi_1}{\partial z} \\ \vdots \\ \rho \chi_{ns-1} Q_{sf} - \rho AD \frac{\partial \chi_{ns-1}}{\partial z} \end{pmatrix} \quad \mathbf{S} = \begin{pmatrix} -\sum_{k=1}^n (\rho Q')_k + \rho S' - \rho \hat{S}' \\ -\rho g \\ -\sum_{k=1}^n (\rho h Q')_k + \rho S'_h - \rho \hat{S}'_h \\ -\sum_{k=1}^n (\rho \chi_1 Q')_k + \rho \chi_1 S' - \rho \chi_1 \hat{S}' \\ \vdots \\ -\sum_{k=1}^n (\rho \chi_{ns-1} Q')_k + \rho \chi_{ns-1} S' - \rho \chi_{ns-1} \hat{S}' \end{pmatrix} \quad (4)$$

where ρ is the mixture density, h enthalpy, χ mass fraction, Q volume flow rate, P pressure, k thermal conductivity, T temperature, D mass diffusion coefficient, Q' jet volumetric entrainment rate per unit length, n the total number of jets, S' and $\hat{S}'$ volumetric source and sink per unit length, S'_h and $\hat{S}'_h$ volumetric energy source and sink per unit length, and ns and sf are subscripts denoting the number of species and stratified ambient fluid, respectively. The readers are recommended to refer to Peterson [6] for a complete description of the large enclosure thermal stratification scaling analysis and the derivation of these ambient governing equations.

To explain why a 1-D method can provide enough information to describe mixing and heat transfer in stratified large volumes, we start with the simplest case. Fig. 1 shows the classical “filling box” problem, which demonstrates major phenomena in stratified mixings [23]. The heating source gives rise to a thermal plume that rises up and spreads over the top of the enclosure, resulting in a stably stratified layer that expands downward with time. The region below the upper stratified layer continues to be at the initial temperature in the enclosure before the onset of the flow. The temperature in the upper heated layer decreases downward from the ceiling to the interface between the upper and lower regions. The flow pattern, the side entrainment into the plume, and the downward motion of the heated upper layer are shown in Fig. 1.

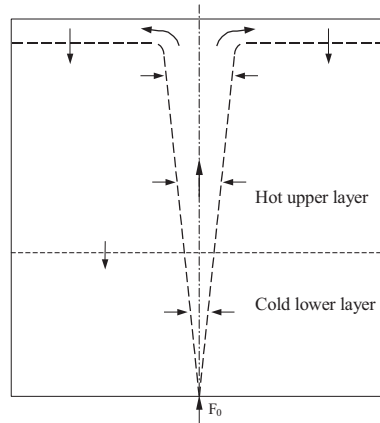


Fig. 1 Sketch of Development of a Stratified Environment Due to a Heat Source, Showing the Motions in the Plume and Environment.

In addition to buoyancy induced plumes, momentum jets, buoyancy jets, steam jets that lead to direct contact condensation, and natural convection boundary layer flows (free wall jets) also are common mixing forces to cause stratification. Large enclosures mixed by buoyant plumes and wall jets can normally be expected to stratify. Furthermore, transition between the

well-mixed and stratified conditions can be predicted [6]. For example, for an injected buoyant jet case, the ambient fluid is stably stratified when

$$\left(\frac{H_{sf}}{d_{bjo}} \right) Ri_{bjo}^{1/3} \left(1 + \frac{d_{bjo}}{4\sqrt{2}\alpha_T H_{sf}} \right)^{2/3} > 1 \quad (5)$$

where H_{sf} is the height of an enclosure, d_{bjo} the diameter of the jet source, $\alpha_T = 0.05$ is the Taylor's jet entrainment constant, and the jet Richardson number (Ri) is given by

$$Ri_{bjo} = \frac{(\rho_a - \rho_o)gd_{bjo}}{\rho_a U_o^2} \quad (6)$$

where ρ_a is the ambient fluid density, ρ_o the source fluid density, g the gravity constant, and U_o the jet source speed.

A jet is simulated with 0-D or 1-D quasi-steady state integral models. Within this paper, a jet should be understood as a generic concept of any steady continuous flow structure in an ambient volume with a dominant flow direction and a length scale much less than the ambient volume's scale. For example, a plume (due to a heat source), a pure jet (due to an initial momentum source) [24], a buoyant jet (due to both buoyancy and momentum [25, 26], a ceiling jet (a jet below the ceiling due to a jet impingement) [27], a wall jet along a wall surface [28], a wall jet due to a normal jet injection, and a wall boundary flow are all taken as jets. All these different jets have a common character: the jet entrains fluid from the ambient volume and finally discharges into the ambient volume. Fig. 2 shows several typical jets.

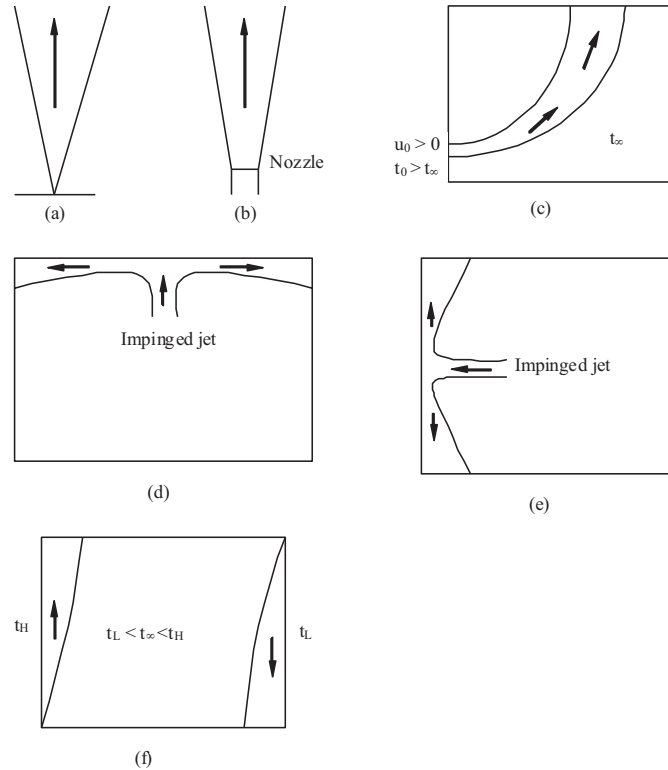


Fig. 2 Typical Jet Types: (a) Pure Plume; (b) Pure Jet; (c) Buoyant Jet; (d) Ceiling Jet; (e) Wall Jet Due to Impinged Jet; (f) Free Wall Jet Due to Wall Boundary Flow.

The BMIX++ code [20, 21, 29, and 22] solves mixing and heat transfer problems in stably

stratified enclosures. The code uses a Lagrangian approach to solve 1-D transient governing equations for the ambient fluid in order to preserve strong gradients in hyperbolically dominated flows. The traditional first order discretization procedures inherently introduce artificial diffusion terms. Typically, these extra diffusion terms impose severe limitations on the maximum size of the computational control volume for the computed solution to be reasonably accurate. The Lagrangian approach [20] eliminates “false diffusion” even with coarse grid and larger time step. The BMIX++ code has a jet model library, including several free jet models for plumes and buoyant jets, a buoyant wall jet model, a ceiling jet model, and two line jet models. In addition, a 1-D transient conduction model for the solid boundaries is included to calculate heat loss through the enclosure walls. Transient heat flux data can be added at the boundaries. Opening models are included to analyze the exchange flow through connections between enclosures. Due to the page limit, detailed physical and numerical models are not described in this paper and readers can refer previous publications for further information [21, 20, and 22].

The BMIX++ code has been successfully validated against multiple benchmark problems, such as stratification in a water tank due to an internal heater with constant power, water tank exchange flow experiment [20], stratification produced by multiple plumes [22], and the UCB large containment mixing experiment, which is composed of a rectangular enclosure with a vertical isothermal cooling wall and a hot air jet injecting [29]. The BMIX++ code also was used to analyze liquid salt pool systems in an advanced high temperature reactor (AHTR) design [22]. Various problems with different combinations can be solved by the BMIX++ code, such as multi-species fluid, variable enclosure cross-section area in vertical direction, multi-enclosures connected with openings, and multiple jets, plumes, and sinks within one enclosure.

3. POOLEX HEAT-UP EXPERIMENT ANALYSIS

The POOLEX experimental facility at Finland was designed to study phenomena relevant to Nordic design BWR suppression pool, including thermal stratification and mixing. Available open reports [3, 17] provide excellent data to validate the BMIX++ code. The POOLEX facility includes a cylindrical stainless steel tank with an outer diameter of 2.4 m and a water pool depth of 2.95 m. The total volume of water was approximately 12 m³ and the blowdown pipe was submerged by 1.81 m. The total duration of the experiment STB-20 was approximately 52 hours. Before the steam blowdown was initiated, the pool water temperature was at 30°C. During the first 4 hours, pool water was heated with steam flow. The steam flow rate (25 to 55 g/s) was slowly reduced during the experiment to make sure that the steam condenses inside the blowdown pipe and the steam-water interface remains close to the blowdown pipe outlet. The pool water stratified strongly during the heating phase. After 4 hours, the temperature of water in the upper part of the pool reached the value of 67°C. The temperature below the blowdown pipe outlet level stayed at the initial value of 30°C. We only simulated the heat-up stage in this paper.

Part of POOLEX experimental data have been simulated with GOTHIC and a commercial CFD tool, Fluent, by Li and Kudinov [17]. The lumped parameter-based GOTHIC simulation results are used in this paper to provide inputs for the BMIX++ and STAR-CCM+ [30] CFD simulations, including the heating rate by condensing steam and heat losses through the pool surface, the cylinder side, and bottom walls.

Fig. 3 shows the schematic for the BMIX++ simulation in which only the water pool is

simulated. BMIX++ uses the point heating source to simulate the heating and mixing source. All boundaries, such as the pool surface and the vessel wall, are simulated as heat flux boundaries. The cross-sectional area variation along the vertical direction is accounted for. The simulation ignores the initial 400 s period without any steam injection and stops at 14,000 s. A time-step of 10 s is used in the simulation. The total simulation time is about 1 minute with a 4-core HP workstation. The total number of control volumes for ambient fluid is about 600. This number is not constant because the BMIX++ code uses the automatic mesh refinement (AMR) technique to capture the sharp interfaces.

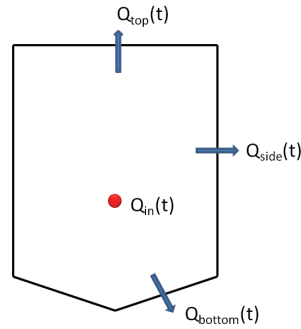


Fig. 3 BMIX++ Simulation Schematic for the POOLEX Heat-up Experiment.

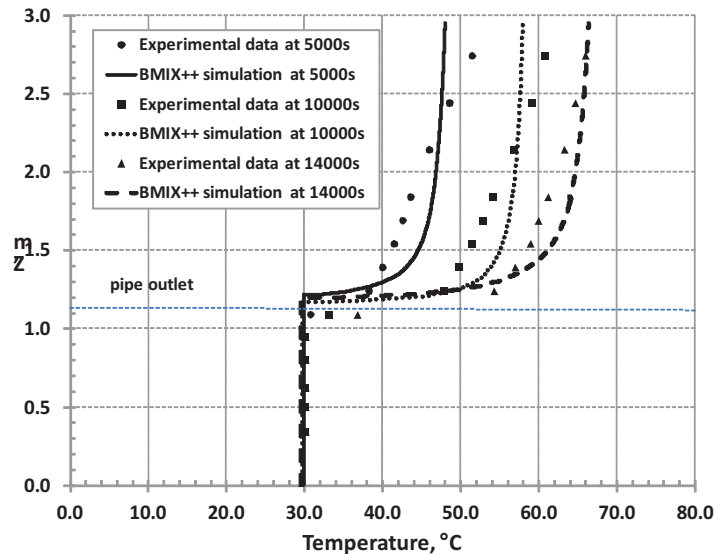


Fig. 4 Temperature Distributions at 5000 s, 10000 s, and 14000 s.

Fig. 4 shows temperature distributions at 5000s, 10000s, and 14000s with discrete points representing experimental data and continuous lines for the BMIX++ simulation results. The simulation results generally agree well with experimental data. The interface location where water temperature begins to rise is near the blowdown pipe outlet. The BMIX++ code predicts interface locations within about 10 cm from the experimental measurement data. The temperature profiles in the upper hotter region predicted by the BMIX++ code have smaller gradients than the experimental data; however, the difference becomes smaller with time. The potential reason for the discrepancy is that the steam may condense along the blowdown pipe in the earlier stages when the pipe wall is still cold. Therefore, some heat is released along the pipe wall instead of the assumption used in our calculations that all heat is released at the pipe outlet. At the end of the heat-up stage, the surface temperature predicted by the BMIX++ code

matches with the experimental data very well. It is noted that the pool surface temperature and warm-cold water interface are the two most important figures of merit for the suppression pool safety analysis. Two different buoyant jet models are used in the calculations: Taylor's vertical buoyant jet model [24] and Schatzmann's generic curved buoyant jet model [25]. Both models generate similar results, with the pool surface temperatures differing by a maximum of 2°C for typical entrainment coefficient values.

CFD simulations using STAR-CCM+ code also were done to provide code comparisons with the BMIX++ results. Fig. 5 shows the geometry and 2-D meshes used for the STAR-CCM+ CFD simulation. The heating source is simulated as a uniformly distributed heat flux boundary along the blowdown pipe. All other boundary conditions are the same as those used for the BMIX++ simulations. The reference run used 3,750 pure quad cells. The regions near the pool surface, around the blowdown pipe outlet, and near other flux boundaries have more dense meshes than other regions. The κ - ϵ turbulence model and implicit unsteady method were used in the calculations. Time step convergence study and comparison with experimental result at 14000 s are shown in Fig. 6. Three time steps at 20 s, 10 s, and 5 s were used. It clearly shows that the results have converged at the 10 s time step and match experimental results very well. Fig. 7 shows 2-D temperature and velocity field information by the STAR-CCM+ CFD simulation at 14000 s. It is noted that strong thermal stratification is formed in the upper region.

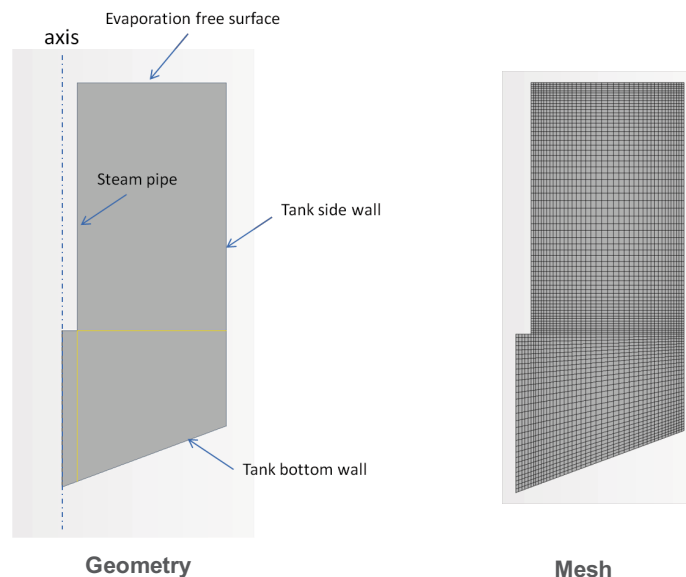


Fig. 5 Geometry and Meshes for CFD Simulation.

Fig. 8 compares the BMIX++ code results with experimental data and CFD (STAR-CCM+ and GOTHIC) simulations at 14000 s. All simulations generally agree well with the experimental data. The converged GOTHIC simulation seems to over-predict the surface temperature. However, in term of computational efficiency, the BMIX++ code is much better for obtaining similarly accurate figures of merit results than other 2-D CFD simulations. The simulation time for the BMIX++ code is about 1 minute versus 1 day for CFD simulations. The difference in simulation time is on the order of 1:1000.

4. CONCLUSIONS

Accurately predicting BWR suppression pool stratification phenomena is important because it affects the peak containment pressure and the NPSHa value, which determines the performance of the emergency cooling pump drawing cooling water back to the reactor core. Current safety analysis codes use 0-D lumped parameter methods to calculate the energy and mass balance in the pool; therefore, they have large uncertainties in the prediction of the scenarios in which stratification and mixing are important. While CFD methods can be used to analyze suppression pool configurations, these methods require fine grid resolution to resolve thin substructures (such as jets and wall boundaries), resulting in a long simulation time. Scaling-based 1-D methods such as those used by the BMIX++ code can give satisfactory results for complex thermal mixing problems under stable stratification condition without resorting to expensive CFD simulations. Therefore, these methods are well suited to couple with advanced system analysis codes. The POOLEX benchmark problem's simulations clearly demonstrate the efficiency advantage of this 1-D based coarse grid method over multiple dimensional CFD methods.

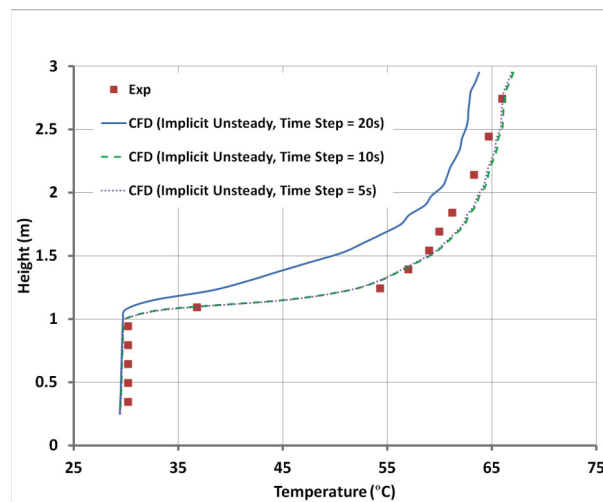


Fig. 6 Time Step Convergence Study for the STAR-CCM+ CFD Simulation (at 14000 s).

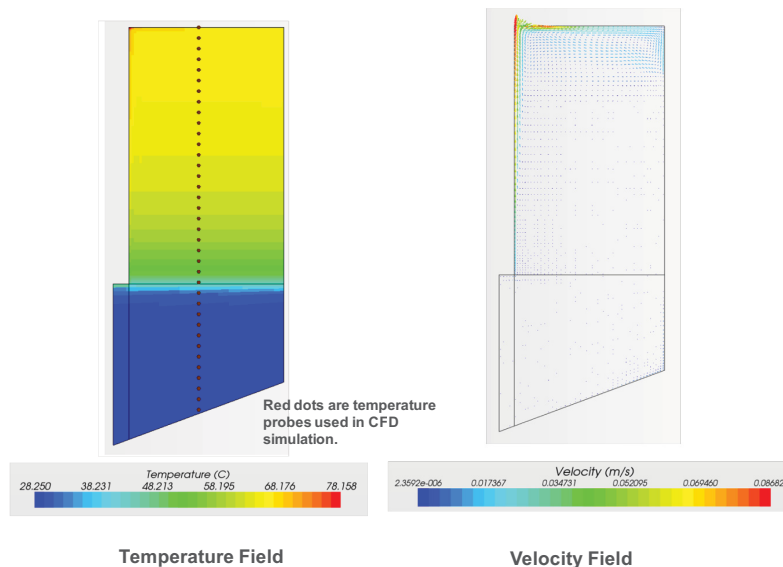


Fig. 7 Temperature and Velocity Fields by the STAR-CCM+ CFD Simulation at 14000 s.

ACKNOWLEDGMENTS

We appreciate Dr. Pavel Kudinov and his team at Royal Institute of Technology, Sweden to give us permission to use their analysis data in the paper. This manuscript has been authored by Battelle Energy Alliance, LLC under Contract No. DE-AC07-05ID14517 with the U.S. Department of Energy. The United States Government retains and the publisher, by accepting the article for publication, acknowledges that the United States Government retains a nonexclusive, paid-up, irrevocable, world-wide license to publish or reproduce the published form of this manuscript, or allow others to do so, for United States Government purposes.

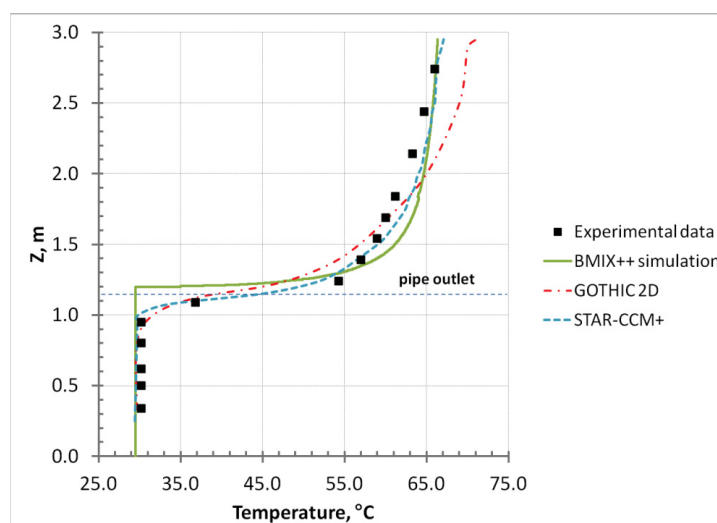


Fig. 8 Comparisons of BMIX++ Code Results with Experimental Data and CFD Simulations.

REFERENCES

- 1 R. E. Gamble et al., "Pressure Suppression Pool Mixing in Passive Advanced BWR Plants," *Nuclear Engineering and Design*, **Vol. 204**, pp 321–336 (2001).
- 2 T. L. Norman, S. T. Revankar, "Jet-plume condensation of steam–air mixtures in subcooled water, Part 1: Experiments," *Nuclear Engineering and Design*, **Vol. 240**, pp 524–532 (2010).
- 3 J. Laine and M. Puustinen, *Thermal Stratification Experiments with the Condensation Pool Test Rig*, Lappeenranta University of Technology, NKS-117, (2006).
- 4 U.S. NRC, *Draft Guidance for Use of Containment Accident Pressure in Determining the NPSH Margin of ECCS and Containment Heat Removal Pumps* (2010).
- 5 O. Auban et al., "Investigation of Large-scale Gas Mixing and Stratification Phenomena related to LWR Containment Studies in the PANDA Facility," *Nuclear Engineering and Design*, **Vol. 237**, pp. 409–419 (2007).
- 6 P. F. Peterson, "Scaling and Analysis of Mixing in Large, Stratified Volumes," *International Journal of Heat and Mass Transfer*, **Vol. 37**, Suppl. 1, pp. 97–106 (1994).
- 7 S. Schwarz et al., "Benchmark on Hydrogen Distribution in a Containment Based on the OECD-NEA THAI HM-2 Experiment," *Nuclear Technology*, **Vol. 175**, pp. 594–603 (2011).
- 8 J. Woodcock et al., "Quantifying the Effects of Break Source Flow Rates on AP600 Containment Stratification," *Nuclear Technology*, **Vol. 134**, pp. 37–48 (2001).
- 9 H. Zhao, P. F. Peterson, "An Overview of Modeling Methods for Thermal Mixing and

- Stratification in Large Enclosures for Reactor Safety Analysis,” *Proceedings of the 8th International Topical Meeting on Nuclear Thermal-Hydraulics, Operation and Safety (NUTHOS-8) in Shanghai, China*, October 10-14 (2010).
- 10 INL, *RELAP5-3D Code Manual Volume I: Code Structure, Systems Models, and Solution Methods*, INEEL-EXT-98-00834, Rev. 2.0, Edition (2002).
 - 11 U.S. Nuclear Regulatory Commission, *TRAC-M/FORTRAN 90 (Version 3.0) Theory Manual*, NUREG/CR-6724 Edition (2001).
 - 12 U.S. Nuclear Regulatory Commission, *MELCOR Computer Code Manuals*, NUREG/CR-6119, Vol. 2, Rev. 3, SAND 2005-5713 (2005).
 - 13 F. E. Dunn et al., “Preliminary Safety Evaluation of the Advanced Burner Test Reactor,” Argonne National Laboratory, ANL-AFCI-172, September 15 (2006).
 - 14 E. Krepper, E. F. Hicken, H. Jaegers, “Investigation of natural convection in large pools,” *Int. J. Heat and Fluid Flow*, **Vol. 23**, pp. 359–365 (2002).
 - 15 A. Gupta et al., “Thermal stratification studies in a side heated water pool for advanced heavy water reactor applications,” *Heat Mass Transfer*, **Vol. 45**, pp. 275–285 (2009).
 - 16 EPRI, *GOTHIC Containment Analysis Program, Version 7.2a-Beta*, Palo Alto, CA. (2005).
 - 17 H. Li, P. Kudinov, *GOTHIC code simulation of thermal stratification in POOLEX facility*, Royal Institute of Technology (KTH), NKS-196, (2009).
 - 18 H. Li, P. Kudinov, W. Villanueva, *Development and Validation of Effective Models for Simulation of Stratification and Mixing Phenomena in a Pool of Water*, Royal Institute of Technology (KTH), NKS-248, (2011).
 - 19 M. Andreani et al., “A benchmark exercise on the use of CFD codes for containment issues using best practice guidelines: A computational challenge,” *Nuclear Engineering and Design*, **Vol. 238**, pp. 502–513 (2008).
 - 20 J. Christensen and P. F. Peterson, “A One-Dimensional Lagrangian Model for Large-Volume Mixing,” *Nuclear Engineering and Design*, **Vol. 204**, pp. 299–320 (2001).
 - 21 H. Zhao, *Computation of Mixing in Large Stably Stratified Enclosures*, Ph.D. dissertation, University of California, Berkeley (2003).
 - 22 H. Zhao and P. F. Peterson, “One-Dimensional Analysis of Thermal Stratification in the AHTR Coolant Pool,” *Nuclear Engineering and Technology*, **Vol. 41**, pp. 953–968, (2009).
 - 23 W. D. Baines and J. S. Turner, “Turbulent Buoyant Convection from a Source in a Confined Region,” *Journal of Fluid Mechanics*, **Vol. 37**, pp. 51–80 (1969).
 - 24 B. R. Morton, “Forced plumes,” *J. Fluid Mech.*, **Vol. 5**, pp. 151–163 (1959).
 - 25 M. Schatzmann, “The Integral Equations for Round Buoyant Jets in Stratified Flows,” *J. Appl. Math. Phys. (ZAMP)*, **Vol. 29**, pp. 608–630 (1978).
 - 26 M. Schatzmann, “A Integral Model for Plume Rise,” *Atmos. Environ.*, **Vol. 13**, pp. 721–731, (1979).
 - 27 Alpert, R. L., “Turbulent Ceiling-Jet Induced by Large-Scale Fires,” *Combustion Science and Technology*, **Vol. 11**, pp. 197–213, (1975).
 - 28 Y. Jaluria, and L. Y. Cooper, “Negatively Buoyant Wall Flows Generated in Enclosure Fires,” *Energy Combust. Sci.*, **Vol. 15**, pp 159–182, (1989).
 - 29 F. Niu et al., “Investigation of Mixed Convection in a Large Rectangular Enclosure,” *Nuclear Engineering and Design*, **Vol. 237**, pp. 1025–1032 (2007).
 - 30 CD-adapco, *User Guide STAR-CCM+ Version 5.06.010* (2010).