Thermal Stratification Analysis for Sodium-cooled Fast Reactors: Development of the 1-D System Model

Cihang Lu[1], Zeyun Wu[1], James Schneider[2], Mark Anderson[2], Liangyu Xu[3], Emilio Baglietto[3], Matthew Bucknor[4], Matthew Weathered[4], Sarah Morgan[1], Sama Bilbao y Leon[5],

[1]*Virginia Commonwealth University:* zwu@vcu.edu
[2]*University of Wisconsin – Madison:* manderson@engr.wisc.edu
[3]*Massachusetts Institute of Technology:*emiliob@mit.edu
[4]*Argonne National Laboratory:* mbucknor@anl.gov
[5]*OECD Nuclear Energy Agency:* *sama.bilbaoyleon@oecd-nea.org*
https://dx.doi.org/10.13182/T31224

INTRODUCTION

The Sodium-cooled Fast Reactor (SFR) is a type of advanced nuclear reactors that are cooled by liquid sodium. Compared to other coolants, as a liquid metal, the liquid sodium has a higher heat capacity and thermal conductivity, and thus can provide an enhanced cooling ability. The fast neutron spectrum in an SFR also grants a better fuel economy because higher burnup can be achieved for both fissile and fertile materials in fast reactors. The SFR was chosen to be one of the six Gen IV reactor designs which represent the future shape of the nuclear energy.

However, several key technology gaps remain to be filled to ensure a safe operation of the SFR before its commercial deployment. The thermal stratification behavior of the liquid sodium, especially in the upper plenum of the pool-type SFR configuration, is one of the key challenges. Stratified layers of liquid sodium with a large vertical temperature gradient could be established in the upper plenum of an SFR during accidental scenarios, including the Protected Loss of Flow (PLOF) and the Unprotected Loss of Flow (ULOF) accidents. The stratified layers are unstable and can result in low-frequency temperature oscillations of fairly large amplitude [1], which can further cause neutronic and thermal-hydraulic instabilities, or result in damages of the structures due to thermal fatigue crack growth.

In order to prevent the occurrence of the thermal stratification, or to mitigate the damages caused, an efficient yet accurate approach to predict it is firstly desired. Both the Computational Fluid Dynamics (CFD) codes and the system codes, such as SAM and SAS4A/SASSYS developed by Argonne National Laboratory (ANL), are able to perform predictions of the thermal stratification phenomena. However, the CFD calculations are too computationally expensive and the system codes are yet able to provide predictions with enough precision.

This paper summarizes recent research efforts on the development of a 1-D system level model with a reasonable fidelity to the prediction of the thermal stratification phenomena in SFR. Comparisons of the preliminary calculation results of the newly developed 1-D model to both the test experimental data and CFD calculations are presented in the current paper.

EXPERIEMNTAL DESIGN

The Thermal Stratification Experimental Facility (TSTF) was developed at the University of Wisconsin-Madison to provide experimental data for the validation of the 1-D model. Fig. 1 gives a diagram of the TSTF. In the experiments, jets of sodium were injected into a pool of sodium from its bottom to mimic the inlet flow to the upper plenum of an SFR. More detailed descriptions of the TSTF can be found in our previous publication [2]. Twelve thermal couples were installed in the test section of the TSTF at six different axial levels for temperature measurements as indicated in Fig. 1.

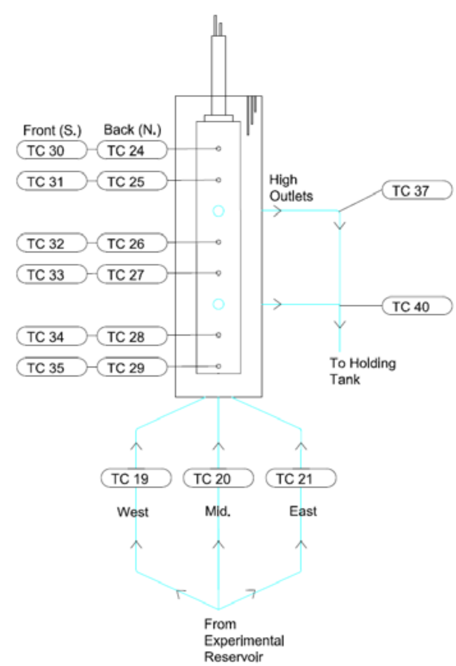

Fig. 1. Positions of the thermal couples in the TSTF.

Two outlets at different levels were designed to examine the effects of the thermal stratification. However, only the high outlet has been used so far to generate experimental data. The temperature measurements, obtained from the 8 thermal couples located lower than the high outlet, were used for the validation of the 1-D model.

CFD MODEL

A CFD model of the test section of the TSTF was developed at the Massachusetts Institute of Technology, and simulations were conducted for several experimental settings of the thermal stratification. The CFD model contained all geometrical details of the test section, including the thermal couples. About five million cells were used to represent the geometry of the test section, as shown in Fig. 2. The CFD model was first built to inform the design of the experimental facility, and then used for the verification of the 1-D system model by comparing the calculation results using both methods.

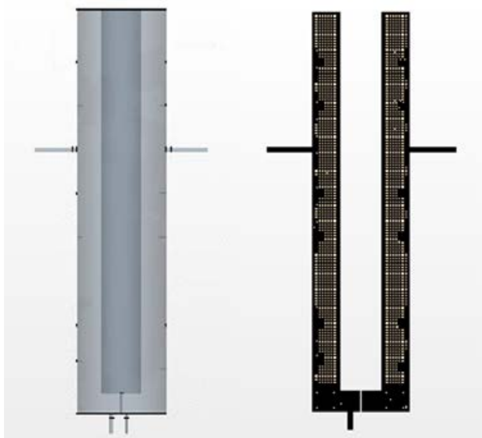

Fig. 2. Geometry and CFD mesh for the TSTF.

1-D SYSTEM LEVEL MODEL

Governing Equations

Following to the work of Peterson [3], the governing field equations for the sodium (ambient fluid) in the upper plenum of an SFR can be simplified as follows, considering the conservation law of mass, momentum, and energy, respectively:

$$A_{sf}(z)\frac{\partial \rho_{sf}}{\partial t}+\frac{\partial(\rho_{sf}Q_{sf})}{\partial z}=\sum_{k=1}^{N_{jet}}\rho_k Q_k' \tag{1}$$

$$\frac{\partial P_{sf}}{\partial z}=-\rho_{sf}g \tag{2}$$

$$A_{sf}(z)\frac{\partial(\rho_{sf}h_{sf})}{\partial t}+\frac{\partial(\rho_{sf}h_{sf}Q_{sf})}{\partial z}-A_{sf}(z)\frac{\partial}{\partial z}\left(k_{sf}\frac{\partial T_{sf}}{\partial z}\right)=\sum_{k=1}^{N_{jet}}\rho_k h_k Q_k' \tag{3}$$

where A_{sf}, Q_{sf} and h_{sf} are the surface area, the vertical volume flow rate and the enthalpy of the ambient fluid. ρ_k, $Q_k{}'$ and h_k are the density, the volumetric entrainment rate and the enthalpy of the k^{th} incoming jet. N_{jet} is the number of all the incoming jets.

By defining the horizontal surface area averaged velocity

$$\bar{u}_z(z)=\frac{Q_{sf}(z)}{A_{sf}(z)} \tag{4}$$

and by combing the equation (1) into (3), we obtain the non-conservative form of the energy conservation equation as follows:

$$\rho_{sf}\frac{\partial h_{sf}}{\partial t}+\rho_{sf}\bar{u}_z\frac{\partial h_{sf}}{\partial z}-\frac{\partial}{\partial z}\left(k_{sf}\frac{\partial T_{sf}}{\partial z}\right)+h_{sf}\frac{1}{A_{sf}(z)}\sum_{k=1}^{N_{jet}}\rho_k Q_k'=\frac{1}{A_{sf}(z)}\sum_{k=1}^{N_{jet}}\rho_k h_k Q_k' \tag{5}$$

By using the thermodynamic relation for enthalpy change $dh=c_p dT$, equation (5) can be written in terms of temperature instead of enthalpy:

$$\rho_{sf}c_p\frac{\partial T_{sf}}{\partial t}+\rho_{sf}c_p\bar{u}_z\frac{\partial T_{sf}}{\partial z}-\frac{\partial}{\partial z}\left(k_{sf}\frac{\partial T_{sf}}{\partial z}\right)=\frac{1}{A_{sf}(z)}\sum_{k=1}^{N_{jet}}(\rho Q')_k\left(h_k-h_{sf}\right) \tag{6}$$

In an SFR, both the ambient fluid in the upper plenum and the forced jets entering the upper plenum are sodium. Therefore, we only considered the case where the entering jets and the ambient fluid are of the same material. When they have temperatures that are close to each other, and all the entering jets are identical, Eq. (6) can be simplified as

$$\rho_{sf}c_p\frac{\partial T_{sf}}{\partial t}+\rho_{sf}c_p\bar{u}_z\frac{\partial T_{sf}}{\partial z}-\frac{\partial}{\partial z}\left(k_{sf}\frac{\partial T_{sf}}{\partial z}\right)=J\left(T_{jet}-T_{sf}\right) \tag{7}$$

where

$$J=\frac{N_{jet}}{A_{sf}}c_{p,jet}\rho_{jet}Q_{jet}' \tag{8}$$

Eq. (7) is the equation that we need to solve for the temperature profile of the ambient fluid, and Q_{jet}' is the parameter that we need to calculate through closure equations.

Numerical Discretization

We used the standard staggered scheme with uniform mesh size for the discretization of the governing equations. Field variables such as density (ρ), pressure (P), enthalpy (h) and temperature (T) are defined at the mesh center, and flow variables such as velocities (u) and volumetric flow rates (Q and Q') are defined at the mesh edge. Control volume approach is used for the spatial discretization, and semi-implicit approach is used for the temporal discretization. The first order spatial derivative is approximated by the upwind scheme and the second order spatial derivative is approximated by the center difference scheme. The discretized form of Eq. (7) for the *mesh-average temperature* of the ambient fluid at the mesh *i* in the time step *n+1* can be written as:

$$\rho_i^n c_{p,i}^n\frac{T_i^{n+1}-T_i^n}{\Delta t}+\rho_i^n c_{p,i}^n\bar{u}_{z,i}^n\frac{T_i^{n+1}-T_{i-1}^{n+1}}{\Delta z}-\frac{1}{\Delta z}k_i^n\left[\frac{T_{i+1}^{n+1}-T_i^{n+1}}{\Delta z}-\frac{T_i^{n+1}-T_{i-1}^{n+1}}{\Delta z}\right]=J_i^n\left(T_{jet}-T_i^n\right) \tag{9}$$

RESULTS

As depicted in Fig. 3, we can consider three different experimental settings to benchmark the 1-D model:

1. The entering jets have a higher temperature (therefore a smaller mass density) than that of the ambient fluid;
2. The entering jets have a lower or equal temperature (therefore a higher or equal mass density) than that of the ambient fluid, and there are obstacles blocking the entering jets at the inlets.
3. The entering jets have a lower or equal temperature (therefore a higher or equal mass density) than that of the ambient fluid, and there is no obstacles in the tank.

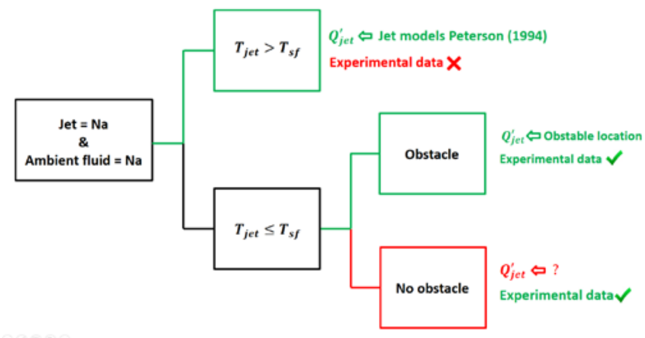

Fig. 3. Three experimental settings considered.

Setting 1: $T_{jet} > T_{sf}$

This situation could occur in an SFR during the ULOF accidental scenario, in which hotter sodium coolant enters the cooler upper plenum from its bottom. After entering the pool, an injected jet would transfer from the forced jet to the buoyancy plume at a certain height. The 1-D model was built with the help of the jet models that are investigated in the work of Peterson [3]. However, we are still acquiring experimental data for this situation.

Setting 2: $T_{jet} \leq T_{sf}$ with obstacles in the pool

This situation could occur in an SFR during the PLOF accidental scenario, in which cooler sodium coolant enters the hotter upper plenum from its bottom. When there are obstacles located close to the inlet of the jet, the entering jet will not be able to rise above the obstacles. The jet could therefore be completely dispersed within the distance between the inlet of the jet and the obstacle.

The experiment considered consisted of three identical sodium jets with a temperature of 200 ℃ entering the test section filled with sodium of 250 ℃. The total volume flow rate of the entering jets was 6 gpm (gallon per minute). In the experiment, we had an Upper Instrumentation Structure (UIS) installed in the test section, the bottom of which was about 5 cm away from the inlet surfaces of the jets. In the 1-D model, we assumed that the injected sodium was evenly dispersed in the ambient fluid within the distance between the bottom of the UIS and the jet inlet surface.

The calculation was completed in less than 0.3 seconds by using the newly developed 1-D model. The temperature of the ambient fluid predicted by the 1-D model at different axial locations are shown in Fig. 4, in comparison with both the CFD predictions and the experimental data. At the beginning of the experiment, a steady state was established in the test section, and the ambient fluid had a uniform temperature. Because of the entering of the cooler sodium, the temperature of the ambient fluid closer to the jet inlets started to decrease. Through the accumulation of the cooler jets dispersed, the temperature of the ambient fluid would eventually converge to that of the entering jets.

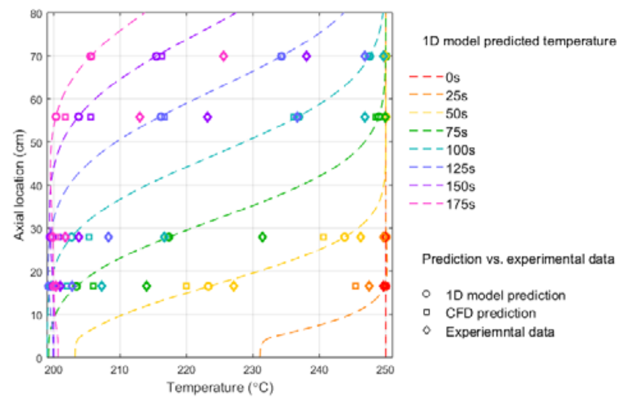

Fig. 4. Comparison of the predicted temperature with experimental data at different axial locations.

The *time-averaged* percentage differences between the predicted axial temperatures and the experimental data were compared, as shown in Fig. 5. Both the 1-D model and the CFD method under-predicted the temperature of the ambient fluid. The CFD method provided slightly better predictions than the 1-D model, but the precisions of both methods were overall very similar at all the axial locations.

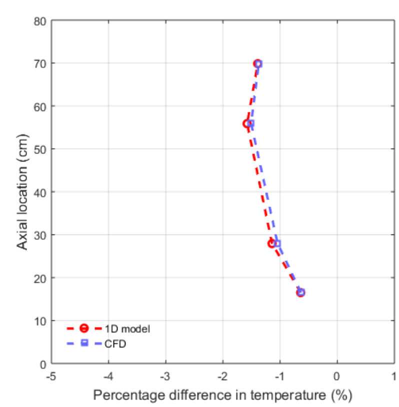

Fig. 5. Time-averaged percentage difference between the predicted temperature and the experimental data.

The temperature of the ambient fluid predicted by the 1-D model as a function of the elapsed time is shown in Fig. 6, in comparisons with both the CFD predictions and the experimental data. It can be seen that the temperature predicted by the 1-D model decreased faster than the experimental data. This is because the mixing of the jet and the ambient fluid took time, while the 1-D model assumed that the dispersion process was instantaneous.

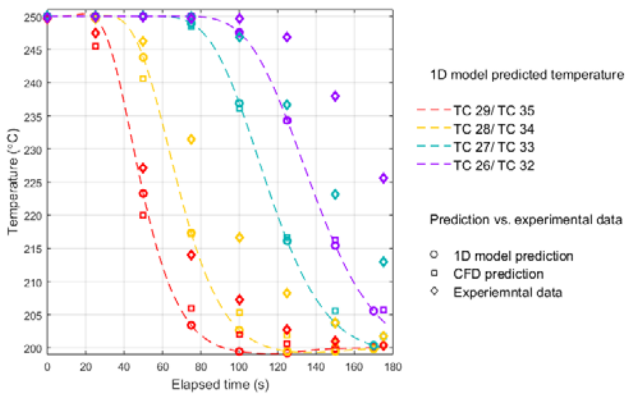

Fig. 6. Comparison of the predicted temperature with experimental data at different elapsed time.

The *axial-location-averaged* percentage differences between the predicted axial temperatures and the experimental data were compared, as shown in Fig. 7. The 1-D model provided slightly better predictions than the CFD method during the first 75 s. The differences between the predicted temperatures and the measurement data became less eminent after 150 s because the temperature of the ambient fluid in the whole test section converged to that of the entering jets. This implies that the axial-location-averaged percentage error of the 1D model is bounded by -2.5% in this experiment.

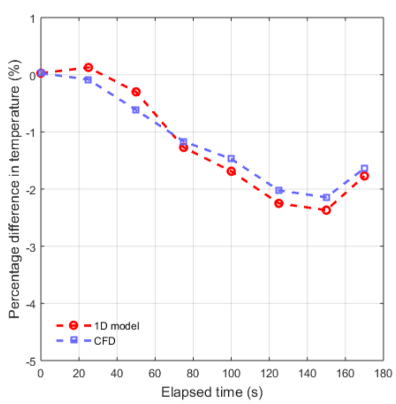

Fig. 7. Axial-location-averaged difference between the predicted temperature and the experimental data.

Setting 3: $T_{jet} \leq T_{sf}$ with no obstacles in the pool

When there is no UIS installed in the tank, although the entering jets would not be blocked, they could only reach a certain height due to both the friction and the gravity force. The Q'_{jet} remained to be predicted in this case. By comparing to the experimental data and the CFD predictions, we found that the 1-D model, with an assumption that the injected sodium was evenly dispersed in the ambient fluid within 75 cm, provided reasonable temperature predictions, as shown in Fig. 8.

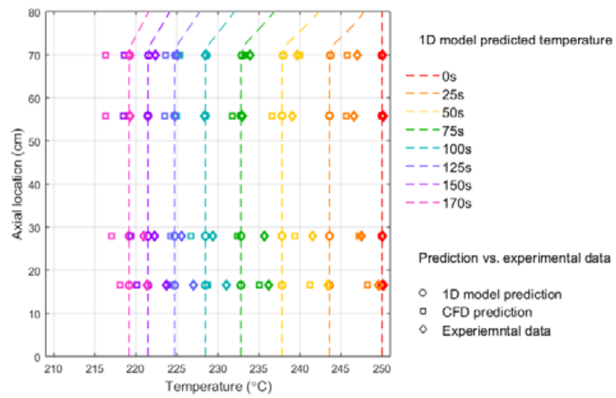

Fig. 8. Comparison of the predicted temperature with experimental data at different axial locations with no UIS.

CONCLUSIONS

A 1-D thermal stratification model based on Berkeley's integral technique was developed and tested on three settings to mimic both ULOF and PLOF accidents in the SFR. For the case that warmer sodium entering a cooler pool, the 1-D model was completed with the Q'_{jet} calculated with the jet models as described in [3]. For the case that cooler sodium entering a warmer pool with the UIS installed, the 1-D model was completed with the Q'_{jet} calculated by assuming that the injected sodium was evenly dispersed in the ambient fluid within the distance between the bottom of the UIS and the jet inlets. The calculation with the 1-D model was completed within 0.3 seconds, and showed good agreements with the CFD predictions and the experimental data. For the case that cooler sodium entering a warmer pool with no UIS installed, experimental data was acquired, but a model of the Q'_{jet} remained to be developed.

ACKNOWLEDGMENT

This project is supported by the Department of Energy Nuclear Energy University Programs (DOE-NEUP).

REFERENCES

1. M. AZARIAN et al., *Nucl. Eng. Des.*, **124**, 417-430 (1990).
2. J. SCHNEIDER et al., Thermal Stratification Analysis for Sodium Fast Reactors. *ICAPP* 2018.
3. P.F. PETERSON, *Int. J. Heat Mass Tran.*, **37**, 97-106 (1994)