CFD Analyses of the Flow Mixing and Stratification in Pool Type SFR

Keenan R. Kresl-Hotz, Mohamed S. El-Genk, Timothy M. Schriener

Institute for Space and Nuclear Power Studies and Nuclear Engineering Department
The University of New Mexico, Albuquerque, NM, 87131
Keenankh02@unm.edu

INTRODUCTION

Pool type Sodium Fast Reactors (SFRs) provide significant performance and safety advantages. They operate at elevated temperatures and slightly below atmospheric pressure due to the low vapor pressure of liquid sodium (Na). This eliminates the need for a reactor pressure vessel. The high operating temperature increases the thermal efficiency of the plant with a superheated steam Rankine cycle (Fig. 1) [2]. The submerged intermediate Na-Na heat exchanger (HEX) in the sodium pool enhances safety and eliminates the necessity for a Na-Na intermediate loop. Furthermore, the large inventory of liquid sodium in the pool provides a large passive storage medium for the decay heat generated after reactor shutdown.

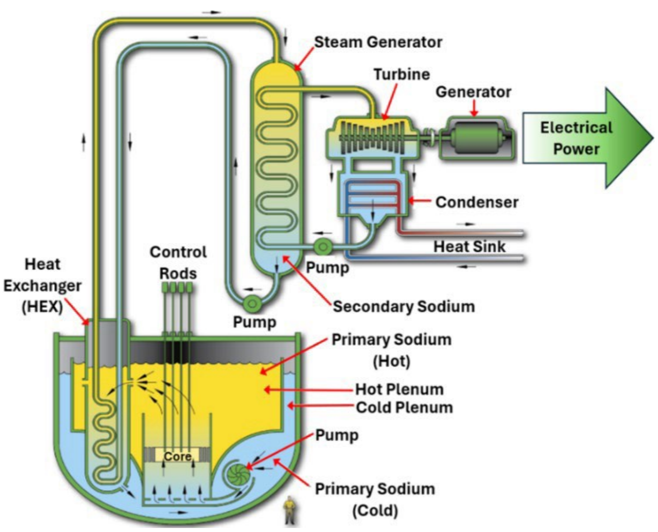

Fig. 1 A Layout of a Pool Type SFR [1].

The hot Na exiting the reactor core enters the submerged Na-Na HEX in the liquid sodium pool and transfers the fission heat removed from the reactor core to the circulating sodium in secondary loop's steam generator (Fig. 1). The produced superheated steam expands through a turbine coupled to an electrical generator for electricity generation. The cooler sodium that exits the HEX enters the sodium pool, then flows through the reactor core to remove the thermal power generated by fission in the nuclear fuel pins.

In the event of a sudden decrease in reactor thermal power or a scram, the temperature of liquid sodium exiting the reactor core decreases. The cooler sodium exiting the core mixes with the hot sodium in the overlying pool and causes stratification of the primary hot sodium in the hot plenum below the free surface. The entrance and the mixing of the hot sodium from the hot plenum into the core may cause thermal and structural stresses. Therefore, it is desirable to investigate the extent of liquid sodium mixing and thermal stratification in the hot plenum of pool type SFRs following a sudden drop in the reactor thermal power.

Researchers at Purdue University have recently investigated flow mixing in the upper plenum of pool-type SFRs, following a sudden decrease in reactor power during a Protected Loss of Power (PLOP) event. The experiments used a scaled liquid Gallium (Ga) experimental facility (GaTE) [3]. The objectives of the present work, performed at the University of New Mexico's Institute for Space and Nuclear Power Studies (UNM-ISNPS) are to perform Computational Fluid Dynamics (CFD) analyses of the liquid Ga experiments to characterize and understand the mixing and stratification of liquid Ga in the upper plenum of the test section (Fig. 2). Previous CFD studies of the experiments [3] used simplified plenum geometry and assumed a rigid Ga free surface. This research investigates modeling the experiment geometry with greater fidelity to improve the comparison of the CFD simulation results with the reported measurements of the liquid Ga's local velocity and temperature.

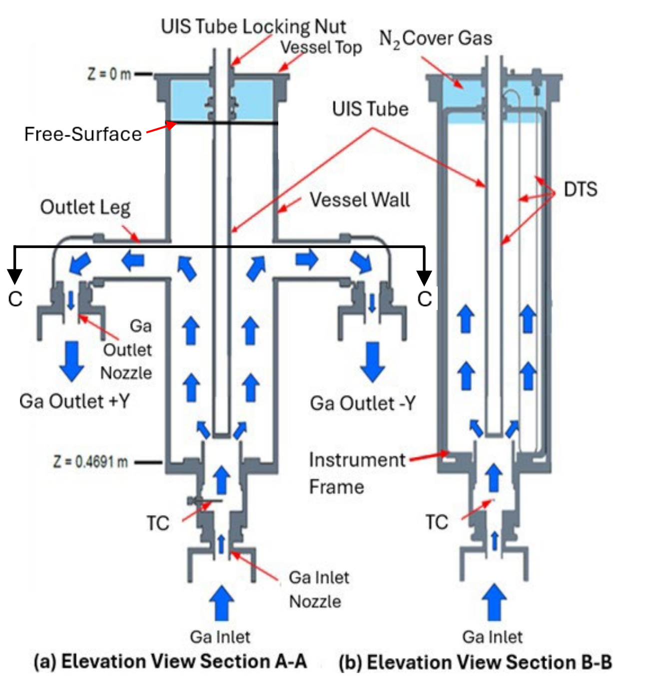

Fig. 2 Cutaway view of the upper plenum of the GaTE experimental facility.

EXPERIMENT SETUP AND CFD ANALYSES

In the Purdue University's forced circulation (GaTE) experimental loop the upper plenum test section (Fig. 2) is a 1/20th scale of that of the liquid sodium pool of the Advanced Burner Test Reactor (ABTR) [3]. The Ga has a low melting point (~303 K) and is compatible with the 316 stainless-steel wall, structure and instrumentation in the upper plenum [3]. The liquid Ga pool in the upper plenum of the test section is covered with a narrow space filled with nitrogen (N_2) cover gas at atmospheric pressure [3].

A fiber optic Distributed Temperature Sensor (DTS) system measures the temperature of the liquid Ga inside the upper plenum at three radial locations using optical frequency domain reflectometry (Fig. 2b) [3]. A type-K thermocouple (TC) measures the temperature of the liquid gallium entering the upper plenum near the inlet nozzle (Fig. 2). An Ultrasonic Doppler Velocimetry (UDV) sensor is mounted to a stage near the top of the instrument frame to measure the axial flow velocity profile at different radial locations within the liquid gallium in the upper plenum. The UDV sensor is not pictured in Figure 2.

The performed CFD analyses calculate the velocity flow field in the Purdue experiment for liquid Ga in the upper plenum. The simulations use the Large Eddy Simulation (LES) [4] turbulence model in the STAR-CCM+ commercial code package [5] to characterize the temporal flow mixing and temperature field in the experiment. The LES model provides complex flow mixing details and the formation of small eddies and vortices [4], at the expense of a high computation cost.

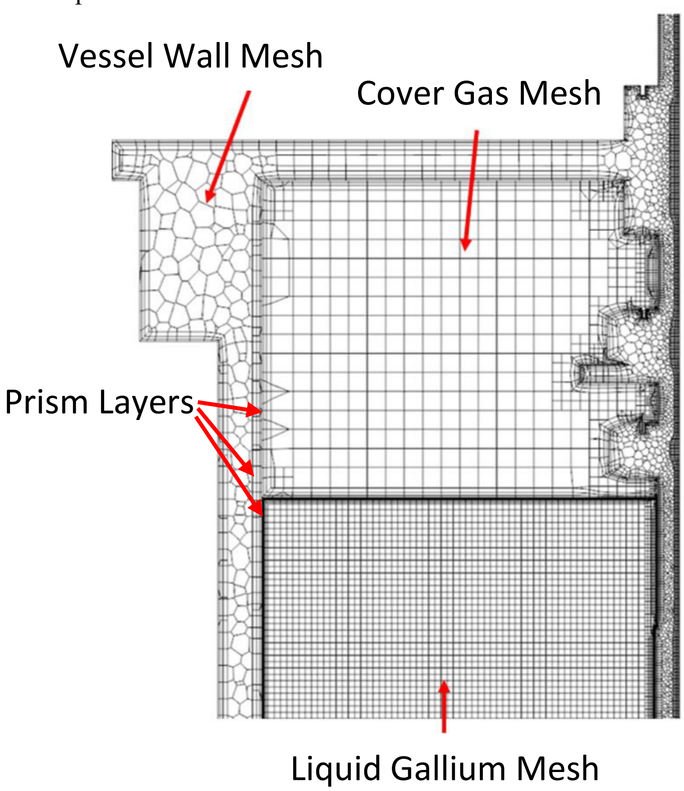

Fig 3. Implemented Numerical Mesh Grid [7].

The CFD analyses employ polyhedral mesh cells in the upper plenum and instrumentation walls, and hexahedral mesh cells in the cover gas and liquid regions. The cover gas is treated as a rigid solid to simplify the calculations. 1.0 mm and 1.5 mm thick liquid and solid regions, respectively, at the common interface comprise prism layers. These are parallel prismatic layers generated near the solid-liquid interfaces with a multiplication factor of 1.3 (Fig. 3). This is to ensure good resolution of the hydrodynamic and thermal boundary layers near the interface.

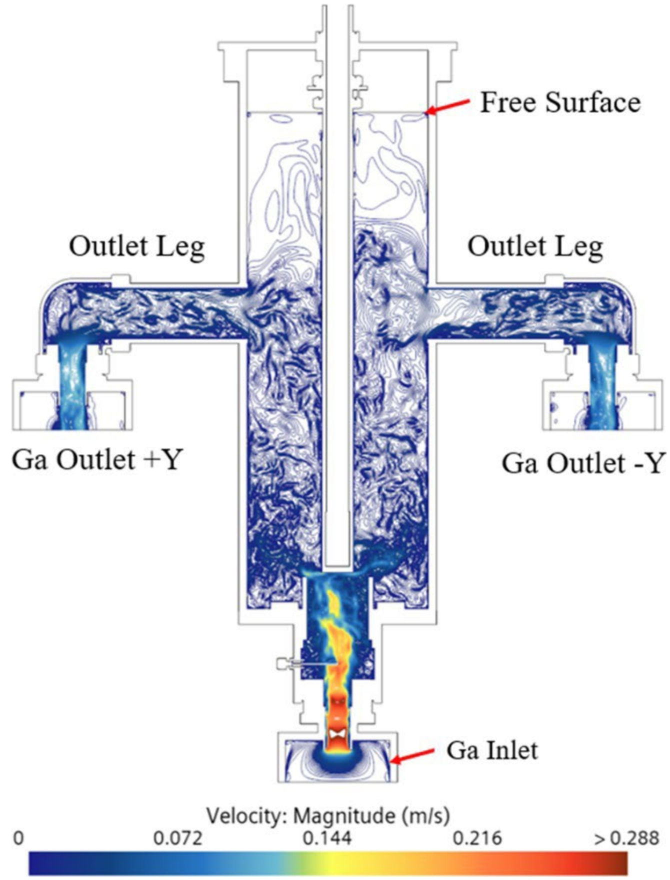

Fig 4. Velocity Contour of Calculated Flow Field.

The performed numerical mesh sensitivity analyses used four grid refinements with increasing total count of the mesh cells [6]: coarse, medium, fine, and finer. Results showed that the fine grid is the best choice for ensuring reliable solution convergence with reasonable computational cost. Therefore, the analyses detailed in the present summary use the fine mesh grid.

The shown CFD calculations are for liquid Ga at an isothermal temperature of 372.15 K, with an inlet velocity of 10 mm/s into the upper plenum, which corresponds to a Reynolds number of 1,770 and total a mass flow rate of 0.3745 kg/s. The pressure of the liquid exiting the upper plenum is specified as atmospheric pressure. The temperatures of the cover gas and the adiabatic vessel wall equal that of the liquid, of 372.15 K. The CFD numerical simulations used an implicit unsteady solver with a timestep of 0.001 seconds.

RESULTS AND DISCUSSION

Velocity contour plots of the isothermal simulation with Gallium inlet velocity of U_{in} = 10 mm/s are provided in Figures 4 and 5. Note the flow pattern of the liquid Ga in the upper plenum of the experiment (Fig. 2). The performed CFD simulations show a chaotic flow field, with the formation of many small-scale eddies. This highlights the ability of the LES turbulence model to capture small flow features contributing to the mixing of liquid Ga within the vessel [5].

Liquid gallium enters the upper plenum through inlet nozzle at the bottom of the entrance section. The small flow area of the inlet nozzle increases the inlet velocity into the liquid pool which intensifies mixing through the formation of liquid jets. For example, at an inlet velocity of U_{in} = 10 mm/s, the entrance velocity into the liquid pool is as much as 0.292 m/s (Fig. 4) The CFD simulations predict chaotic turbulent flow and intensive mixing of liquid gallium in the lower section of the upper plenum (Fig. 4). The formation of many small eddies intensifies mixing in the upper plenum before liquid gallium exits through the outlet legs, which represent the entrance to the Na-Na HEX in the pool of an SFR (Figs. 2 and 5).

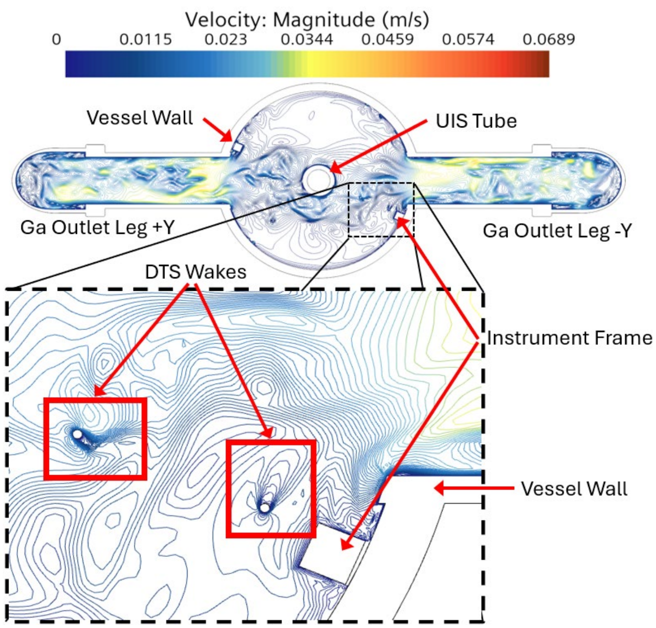

Fig 5. Section View C-C of Velocity Contour of Calculated Flow Field Showing DTS wakes.

Bindra et al [3] reported experimental measurements for an inlet Ga velocity of 60 mm/s of the time-averaged velocity in the upper plenum for axial positions above the horizontal outlet legs with a spatial resolution of 0.97 mm. Four separate axial time-averaged velocity profile measurements were reported by [3] for each of three radial UDV locations r = 36, 47, and 58 mm. The calculated time average velocity profiles determined from a CFD simulation with a Ga inlet velocity of U_{in} = 60 mm/s are generally bounded by the four reported experimental time-average velocity profiles at each radial location. The CFD results agree with the four experimental measurement sets to within +16.2 to -10.4 mm/s for the r = 36 mm probe location, +13.5 to -7.4 mm/s for the r = 47 mm probe location, and +14.0 to -12.8 mm/s for the r = 58 mm probe location.

CONCLUSION

The performed isothermal CFD simulations of the flow field in the upper plenum of Purdue experiment reveal chaotic flow patterns. This includes the formation of many small-scale eddies near the inlet nozzle. The jetting of the liquid exiting the nozzle into the upper plenum aids mixing below the outlet legs. Above the outlet legs, little flow mixing occurs as the liquid is stagnant. The results highlight the difference of the liquid mixing that occurs in the upper plenum above and below the outlet legs. The performed CFD calculations for U_{in} = 60 mm/s have shown some agreement with the reported experimental velocity values. Future work will perform simulations with different inlet velocities to compare with the reported experimental values.

The planned transient simulations will model the PLOP experiments, injecting cold Ga to investigate the formation of thermal stratification of the liquid in the upper plenum. In addition, future work will investigate the effects of simulating the motion of the Ga free surface using an Eulerian method on the mixing and thermal stratification. The CFD mixing and stratification results will be compared to the reported PLOP experiments and previously conducted CFD analyses with assumed rigid liquid free surface [3].

NOMENCLATURE & ACRONYMS

ABTR = Advanced Burner Test Reactor
CFD = Computational Fluid Dynamics
DTS = Distributed Temperature Sensor
Ga = Gallium
GaTE = Gallium Thermal-hydraulic Experiment
HEX = Heat Exchanger
ISNPS = Institute for Space and Nuclear Power Studies
LES = Large Eddy Simulation
Na = Sodium
N_2 = Nitrogen
PLOP = Protected Loss of Power
SFR = Sodium Fast Reactor
TC = Thermocouple
UDV = Ultrasonic Doppler Velocimetry
UIS = Upper Instrumentation Structure
UNM = University of New Mexico

ACKNOWLEDGMENTS

This research is supported by a DOE NEUP IRP grant through a subaward from the City College of New York to the University of New Mexico. We also thank Dr. Hitesh Bindra and Broderick Sieh at Purdue University, for providing the CAD geometry for the GATE facility and information on the performed experiments. We would also like to thank the UNM Center for Advanced Research Computing, supported in part by the National Science Foundation, for the computing resources used in this work.

REFERENCES

1. U.S. Department of Energy, Sodium-Cooled Fast Reactor [Fact Sheet], (2021).
2. INTERNATIONAL ATOMIC ENERGY AGENCY, “Design Features and Operating Experience of Experimental Fast Reactors,” *IAEA Nuclear Energy Series,* NP-T-1.9, Vienna (2013).
3. H. BINDRA, et al., “A Computational-Experimental Study to Simulate Mixing and Thermal Stratification in SFRs,” NEUP 16-10579, Final Report, (2020).
4. P. SAGAUT, *Large Eddy Simulation for Incompressible Flows: An Introduction*., 3rd Ed., Scientific Computation Series, Springer-Verlag, Berlin, (2006).
5. Siemens Digital Industries Software. Simcenter STAR-CCM+, version 2306.0001, Siemens (2023).
6. K.R. KRESL-HOTZ, et al. “Numerical Mesh Refinement for flow mixing in Pool Type SFR,” *American Nuclear Society Student Conference*, Albuquerque, NM, Apr. 3-5, (2025).