# Effect of Numerical Mesh Refinement for Simulating Flow Mixing in Pool Type SFRs

**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) offer many performance and safety advantages. They operate at elevated temperatures and slightly below atmospheric pressure owing to the low vapor pressure of liquid sodium (Na). This eliminates the need for a heavy 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 need for a Na-Na intermediate loop. Furthermore, the large inventory of liquid sodium in the pool provides a large medium for the passive storage of decay heat after reactor shutdown.

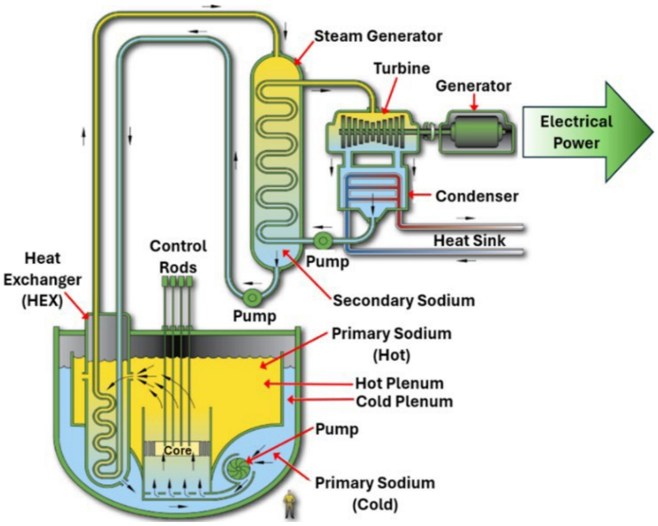


Fig. 1 A Layout of a Pool Type SFR [1].

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

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

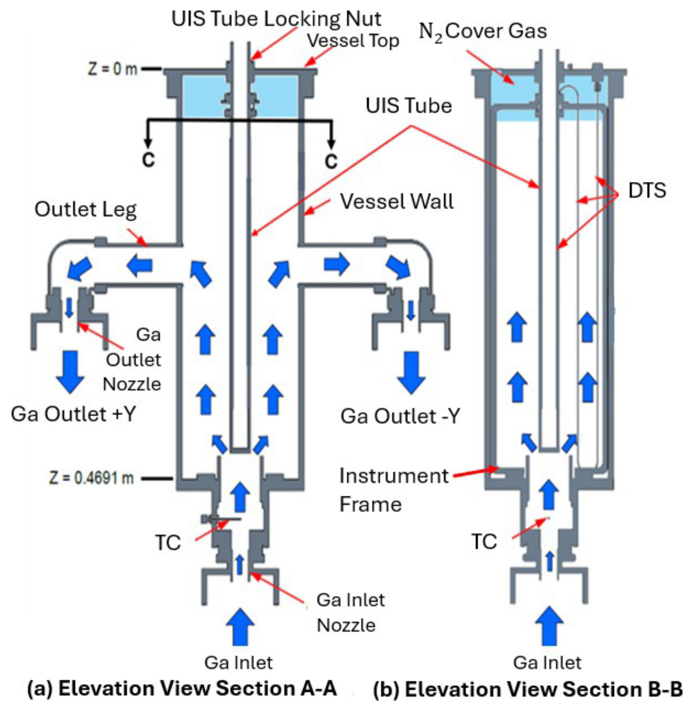


Fig. 2 Cutaway view of the upper plenum of the GaTE experimental facility.

Researchers at Purdue have recently investigated flow mixing in pool-type SFRs, following a sudden decrease in reactor power, using a scaled down liquid Gallium (Ga) experimental facility (GaTE) [3], as it is impractical to do so with an actual reactor. The objective of the present work performed at the University of New Mexico's Institute for Space and Nuclear Power Studies (UNM-ISNPS) is to conduct Computational Fluid Dynamic (CFD) analyses of the Purdue experiment to better understand and simulate the reported mixing and stratification of liquid Ga in the upper plenum of the test section (Fig. 2). The performed CFD analyses employ the STAR-CCM+ commercial code package [4] to characterize the temporal flow mixing and temperature field in the experiments. The performed CFD simulations use the Large Eddy Simulation (LES) turbulence model [5]. It is better suited for capturing the formation and extent of the formation of eddies, swirling vortices, and stratification in the liquid pool. LES models, however, require refined numerical mesh grids to resolve the turbulent eddies in the flow.

This study examines the impact of numerical mesh refinement on the CFD simulation results of the liquid Ga flow mixing in the upper plenum of the Purdue experiment. The focus is to determine the effect of increasing the numerical mesh grid refinement in the CFD simulation on the solution convergence and the computational cost. The analyses examined four numerical mesh grid refinements with increasing total cell counts, by decreasing the sizes of mesh grid cells in the bulk liquid and in the prism layers next to solid surfaces. The calculated Grid Convergence Index (GCI) [6] helps quantify the effect of mesh grid refinement on the convergence of the solution for predicting the flow velocities and the pressure losses in the experiments.

## EXPERIMENT SETUP AND CFD ANALYSES

The Purdue forced circulation (GaTE) experimental loop comprises an upper plenum of liquid Ga as a surrogate for liquid sodium, with a layer of nitrogen ($N_2$) cover gas (Fig. 2). The plenum test section is a 1/20th scale of that of the liquid sodium pool the Advanced Burner Test Reactor (ABTR) [3]. The liquid Ga has a low melting point (~303 K) and is compatible with the 316 stainless-steel walls, 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 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 Ga in the upper plenum. The UDV sensor body, not pictured in Figure 2, is not included in the present CFD analyses.

The polyhedral meshing model meshes the walls of the upper plenum and instrumentation (Fig. 3). The prism layer mesher generates parallel prismatic layers near the solid-liquid interfaces with a multiplication factor of 1.3 to ensure resolution of the hydrodynamic and thermal boundary layers near the interface. Other numerical mesh grid parameters are shown in Table 1.

The GCI quantifies the effect of numerical mesh grid refinement on the solution convergence in the present CFD analyses [6]. It utilizes the relative error of some flow parameter in the formulation. The calculated parameters in the present CFD simulations are the liquid gallium inlet pressure, relative to the outlet pressure, assumed atmospheric, and the average outlet velocity and the mass flow rates of liquid gallium in the experiments. Based upon the GCI values calculated, a mesh refinement will be chosen to be used for simulations of the experiment. The CFD temperature and velocity results from these simulations will be compared to those from the DTS and UDV results from the experiment conducted at Purdue [3].

Fig. 3 presents an image of the medium numerical mesh grid in the present CFD analyses with the nitrogen cover gas is in contact with the free surface of the liquid Gallium in the upper plenum. The core mesh cells in the nitrogen cover gas are coarser than those in the liquid Ga pool in the upper plenum, which is true for all other numerical mesh refinements investigated (Fig. 3).

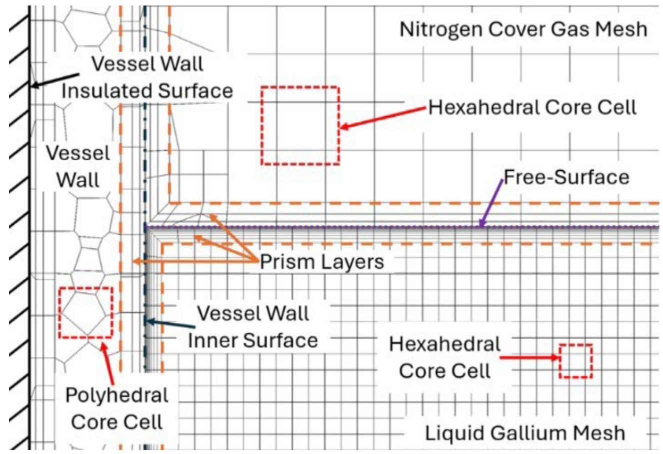


Fig. 3 Numerical Mesh grid in liquid Gallium and near solid surfaces.

The GCI quantifies the effect of numerical mesh grid refinement on the solution convergence in the present CFD analyses [6]. It utilizes the relative error of some flow parameter in the formulation. The calculated parameters in the present CFD simulations are the liquid gallium inlet pressure, relative to the outlet pressure, assumed atmospheric, and the average outlet velocity and the mass flow rates of liquid gallium in the experiments. Based upon the GCI values calculated, a mesh refinement will be chosen to be used for simulations of the experiment. The CFD temperature and velocity results from these simulations will be compared to those from the DTS and UDV results from the experiment conducted at Purdue [3].

## RESULTS AND DISCUSSION

Results presented in Fig. 4 and Table 2 show the effect of increasing the numerical mesh refinement on the calculated value of the GCI. Increasing the numerical mesh grid refinement decreases the conversion uncertainties of the results, and hence the GCI value. For the medium grid, the GCI is much higher than for the fine and finer grid refinements (Fig. 4). The difference in GCI values between the fine and finer mesh grids is negligible, whereas the computational cost is fourfold higher. Because of the small difference in GCI values compared to the large decrease in computational cost, future analyses will use fine grid refinement, without affecting the results. Using this grid refinement, the average outlet velocity from the upper plenum agrees within +1.6% of experimental values, the outlet mass flow rate within +0.8%, and the total pressure loss within +0.07%. Fig. 5 compares images of the calculated velocity field in the upper plenum of the Ga experiment using the coarse and fine mesh grids. The calculated flow field using the fine grid is more chaotic as the LES turbulence model better resolves the forming mixing eddies in the flow.

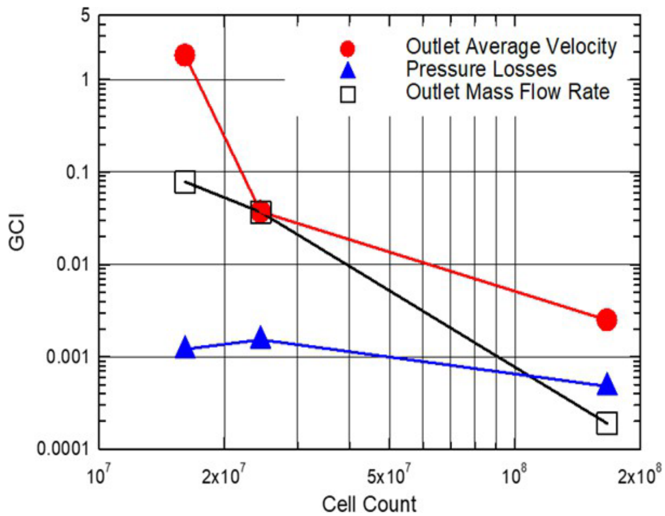


Fig. 4 Comparison of the effect of the numerical mesh refinement on the GCI.

The results also demonstrate that the coarse grid is incapable of resolving all primary eddies of liquid mixing in simulation [5]. Such a limitation is undesirable since these eddies carry most of the turbulent kinetic energy. This effect is very pronounced when comparing the calculated flow fields using the coarse and the fine mesh Grids (Fig. 5). Small eddies form as the liquid Ga emerges as a jet from the entrance nozzle into the upper plenum before entering the outlet flow legs. The coarse mesh does not adequately capture the extent of the liquid flow mixing in the upper plenum, which is captured in the simulation with the fine grid. The resolved liquid mixing and the forming turbulent eddies in the performed simulations using the fine and finer mesh grids are similar, supporting the selection of the fine mesh grid refinement considering the large decrease in the computation cost.

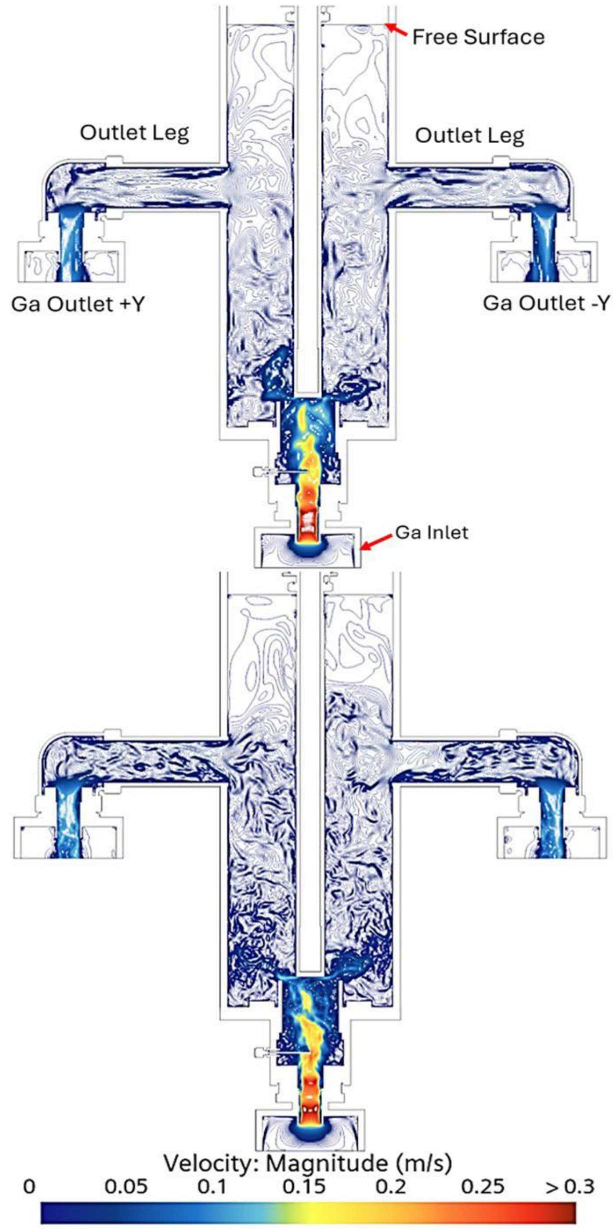


Fig. 5 Comparison of the Calculated Flow Field and Velocity Contours using Coarse (Top) and the Fine (Bottom) Mesh Grids.

Table I. Numerical Mesh Grid Parameters.

| Mesh Grid | Domain | Base Cell Size (mm) | No. of Prism Layers | Prism Layer Total Thickness (mm) | Cell Count (Millions) | Total Cell Count (Millions) |
|---|---|---|---|---|---|---|
| Coarse | Liquid Gallium | 2 | 4 | 1 | 5.44 | 9.16 |
| | Vessel Wall | 4 | 2 | 1.5 | 3.38 | |
| | Cover Gas | 4 | 2 | 1.5 | 0.33 | |
| Medium | Liquid Gallium | 1.2 | 6 | 1 | 11.52 | 16.09 |
| | Vessel Wall | 3.3 | 3 | 1.5 | 4.16 | |
| | Cover Gas | 3.3 | 3 | 1.5 | 0.4 | |
| Fine | Liquid Gallium | 1 | 8 | 1 | 18.39 | 24.47 |
| | Vessel Wall | 3 | 3 | 1.5 | 5.55 | |
| | Cover Gas | 3 | 3 | 1.5 | 0.53 | |
| Finer | Liquid Gallium | 0.5 | 12 | 1 | 97.89 | 116.53 |
| | Vessel Wall | 1 | 4 | 1.5 | 16.78 | |
| | Cover Gas | 1 | 4 | 1.5 | 1.86 | |

Table II. Effect of Mesh Grid Refinement on the Calculated GCI Value Parameters.

| Mesh Grid | Total Cell Count (millions) | Relative Refinement Ratio | Computational Time (CPU Days) | Avg. GCI |
|---|---|---|---|---|
| Coarse | 9.2 | 1 | 188.9 | - |
| Medium | 16.1 | 1.21 | 331.8 | 0.5543 |
| Fine | 24.4 | 1.39 | 453.7 | 0.0221 |
| Finer | 116.5 | 2.33 | 1719.9 | 0.0029 |

## CONCLUSION

The performed mesh grid sensitivity analyses demonstrate the effectiveness of the LES turbulence model for detailed characterization of the flow field and the mixing of the liquid Gallium in the upper plenum of experiments simulating a Protected Loss of Power (PLOP) event. Results show the fine mesh grid achieves good convergence at just one-fourth the computational cost of the finer grid. Therefore, future simulations will be performed using the fine mesh grid to explore the use of an Eulerian method to assess the effect of assuming a nonrigid free surface for the liquid pool on the CFD analyses results [3]. The CFD mixing and stratification results will be compared to those reported in the PLOP experiments at different Ga inlet flow velocities [3].

## NOMENCLATURE & ACRONYMS

ABTR = Advanced Burner Test Reactor
CFD = Computational Fluid Dynamics
CPU = Central Processing Unit
DTS = Distributed Temperature Sensor
Ga = Gallium
GaTE = Gallium Thermal-hydraulic Experiment
GCI = Grid Convergence Index
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
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. Siemens Digital Industries Software. Simcenter STAR-CCM+, version 2306.0001, Siemens (2023).
5. P. SAGAUT, *Large Eddy Simulation for Incompressible Flows: An Introduction*., 3rd Ed., Scientific Computation Series, Springer-Verlag, Berlin, (2006).
6. P.J. ROACHE, "Perspective: A Method for Uniform Reporting of Grid Refinement Studies," *Journal of Fluids Engineering*, **116**, 405 (1994)