Comparative Study Between RANS and LES Simulation of HTTF Reactor

Jae-Ho Jeong[a*], Myeong-Jin seo[a], Han-Seop Song[a]
[a]*Department of Mechanical Engineering, Gachon University, 1332, Seongnam-daero, Sujeong-gu, Seongnam-si, Gyeonggi-do 13306, Republic of Korea*
[*]*Corresponding author: jaeho.jeong@gachon.ac.kr*

***Keywords : CFD, HTGR, RANS, LES**

1. Introduction

The High-Temperature Gas-cooled Reactor (HTGR) is one of the proposed six Next Generation Nuclear Power Plants (NGNP) by the Generation IV International Forum (GIF). Notably, the HTGR stands out due to its capability to achieve helium coolant temperatures exceeding 700°C. This advantageous feature makes it particularly suitable for integration with high-temperature industries and hydrogen production processes. The subsequent evolution of the HTGR, known as the Very High-Temperature Reactor (VHTR), targets even higher exit temperatures, setting the goal at 950°C. The benefits of such elevated exit temperatures, especially for efficient mass hydrogen production, are being reevaluated. Addressing numerous performance and safety concerns associated with thermal hydraulics in the VHTR, as confirmed by a comprehensive report published in 2002, is of paramount importance for the successful fulfillment of its mission [1].

Supported by the United States Department of Energy (US-DOE), the High-Temperature Test Facility (HTTF) at Oregon State University was designed to experimentally investigate the transient behavior of a high-temperature prismatic gas-cooled reactor [2]. This Integrated Effect Test (IET) facility represents a 1/4-scale model of the General Atomic MHTGR [3]. The facility serves as a platform for the validation and verification (V&V) of codes that integrate system thermal-hydraulic codes, computational fluid dynamics (CFD) codes, and system-CFD coupling. It provides experimental data crucial for verifying and validating the performance of these codes. The HTTF plays a significant role in enhancing the understanding of high-temperature gas-cooled reactor behavior through scaled experiments. By providing data that aids in individual modeling tasks, the facility contributes to the advancement of thermal-hydraulic and computational fluid dynamics simulations, thereby supporting the development and safety assessment of advanced reactor designs.

In this study, computational fluid dynamics (CFD) analyses were conducted using the Reynolds-Averaged Navier-Stokes Simulations (RANS) and Large Eddy Simulation (LES) models for the lower plenum of the High-Temperature Test Facility (HTTF), a modular high-temperature gas-cooled 1/4-scale test facility. The focus of the analysis centered around the attachment pillars of the thermal striping in the lower plenum, involving mixing high-temperature and low-temperature helium gases. Through this comparison, the temperature fluctuation characteristics were evaluated.

2. Numerical Methodology

2.1 Analysis modeling

The HTTF experimental device is a cooling test apparatus equipped with a high-temperature heating system, and its general configuration is depicted in Fig. 1. This experimental setup was developed to investigate the thermal properties of materials under high temperature and pressure conditions. Helium is used as the cooling gas, which is supplied to the lower plenum for experimentation and then directed to the T-junction through an exit duct. The inflow of the cooling gas is divided into a total of 5 groups, with each group entering the lower plenum with varying temperatures and mass flow rates. This enables the investigation of heat transfer characteristics under diverse conditions within the experimental environment. The introduced helium exits the system through the exit duct, completing the experimental process. Through the above-mentioned diagram, the configuration of the experimental device and the flow path of the cooling gas can be comprehended. This system allows for obtaining experimental data under various conditions, providing new insights into heat transfer phenomena.

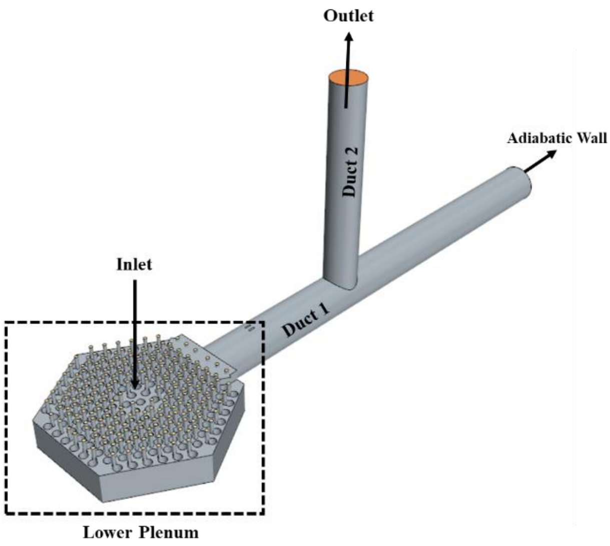

Fig. 1. Schematic of HTTF simulation domain

2.2 Boundary conditions and grid system

The boundary conditions for the HTTF experimental device are presented in Table 1. Inlet conditions are categorized into five distinct groups, each entering with varying flow rates. The exit conditions are set as pressure outlets, with the exit mass flow rate matching the total inlet mass flow rate to the experimental apparatus. Inlet walls, rake, and adiabatic walls are specified as adiabatic conditions, while the remaining walls are maintained at constant temperatures. A no-slip condition was applied to all walls for the computational fluid analysis. The grid for computational fluid analysis was generated using STAR-CCM+, and it is depicted in Figure 2. The grid consists of approximately 80 million hexahedral cells.

2.3 Numerical Physics

Numerical analysis techniques for turbulent flow can be categorized into three types: DNS (Direct Numerical Simulation), LES (Large-Eddy Simulation), and RANS (Reynolds-Averaged Navier-Stokes). DNS is employed to accurately resolve the behavior of the entire turbulent flow field in the presence of various scales of vortices. This requires grid sizes smaller than the smallest turbulence scale and time steps smaller than the smallest time scale of turbulence fluctuations. LES involves directly simulating the larger coherent turbulent structures using the computational grid while modeling the smaller turbulent structures using subgrid scale (SGS) models. The Smagorinsky model, initially proposed by Smagorinsky [4], is one such example. RANS models utilize time-averaging techniques to average out all unsteadiness and approximate them with engineering models [5]. Compared to DNS and LES, RANS offers a lower resolution of flow fields, but it finds extensive application in engineering practice due to its reduced computational demands. In this study, for capturing the turbulent behavior in the HTTF experimental device, RANS models (k-ε) and LES models were employed using the commercial software STAR-CCM+.

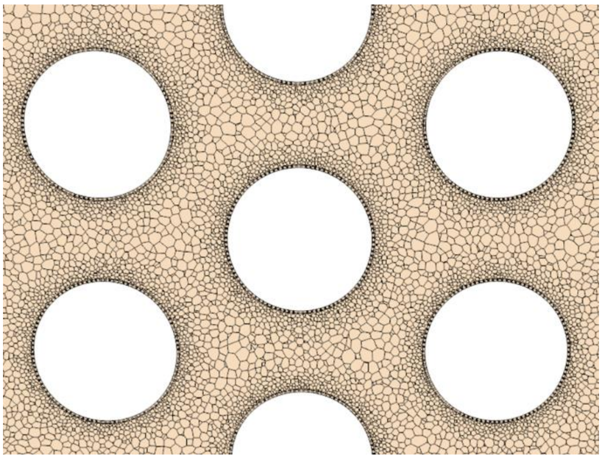

Fig. 2. Mesh cell size contour in the 50% height sectional

The solver settings for the turbulence models used are outlined in Table 2 and Table 3.

Table 1: Boundary Condition

	Name	Mass flow rate [kg/s]	Total Temperature [K]
Inlet	Inlet 1	1.30E-3	562.22
	Inlet 2	9.83E-3	561.84
	Inlet 3	1.48E-2	541.34
	Inlet 4	1.54E-2	512.03
	Inlet 5	4.67E3	471.64
Outlet	Name	Gauge Pressure [Pa]	Static temperature [K]
	outlet	110486.05	504.25
Wall	Name	Condition	Temperature
	Duct Wall 1	No-slip	416.17
	Duct Wall 2		309.32
	Lower plenum side		435.71
	Lower plenum Bottom		476.88
	Lower Plenum Top		565.68
	Column Walls		518.26
	Extruded Inlet Wall		Adiabatic
	Rake		Adiabatic

Table 2: RANS model solver setup for HTTF lower plenum

Simulation type	3D, steady
Turbulence model	k-ε
Convection scheme	Bounded-central
Temporal discretization	Second-order

Table 3: LES model solver setup for HTTF lower plenum

Simulation type	3D, Implicit-unsteady
Turbulence model	LES
SGS model	WALE
Convection scheme	Bounded-central
Time step size [s]	1.0E-4
Temporal discretization	Second-order
Inner loop iteration	10

2.4 Turbulence Model Governing Equations

2.4.1 RANS *k-e* model

The k-ε turbulence model simulates turbulence by accounting for turbulence generation term k and turbulence dissipation rate ε. Additionally, it efficiently incorporates the boundary layer around wall regions using wall functions, making it computationally more efficient compared to k-ω and k-ω SST turbulence models. The k-ε turbulence model can be represented as shown in Equations (1) and (2):

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}(\rho k u_j) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b - \rho\epsilon - Y_M + S_k \; (1)$$

$$\frac{\partial}{\partial t}(\rho \epsilon) + \frac{\partial}{\partial x_j}(\rho \epsilon u_j) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial \epsilon}{\partial x_j}\right] + \rho C_1 S\epsilon - \rho C_2 \frac{\epsilon^2}{k+\sqrt{\nu\epsilon}} + C_{1\epsilon}\frac{\epsilon}{k}C_{3\epsilon}G_b + s_\epsilon \; (2)$$

In the context of the turbulence model equations G_k represents the production of turbulence kinetic energy due to mean velocity gradients. turbulent kinetic energy, G_b accounts for the generation of turbulence kinetic energy due to buoyancy effects, Y_M reflects the influence of compressibility on the expansion of fluctuating quantities in compressible flows, S_k and S_ε are user-defined functions that allow customization of turbulence generation and dissipation rates.

The k-ε turbulence model encompasses the generation and dissipation of turbulence kinetic energy while considering the influence of various factors such as velocity gradients, pressure, and boundary conditions. It efficiently captures turbulence characteristics and offers practical computational advantages, making it suitable for a range of engineering applications.

2.4.2 LES WALE model

The LES WALE (Wall-Adapting Local-Eddy Viscosity) subgrid scale model represents a more contemporary approach to subgrid scale modeling, utilizing a novel formulation for the velocity gradient tensor in its formulation. It addresses some limitations of previous models by introducing a new framework for subgrid scale modeling. Similar to the Smagorinsky subgrid scale model, the model coefficient C_w is not universally defined, which can be considered a drawback.[6] However, the WALE model has the advantage of not requiring near-wall damping, as it automatically provides accurate scaling near the walls. The distinguishing feature of the WALE model lies in its use of a refined velocity gradient tensor formulation, contributing to its improved accuracy in representing turbulent flows. Additionally, the absence of the need for wall damping and its inherent ability to provide accurate scaling near walls contribute to its practicality and reliability. The Wale subgrid scale model provides the following mixing length type formula for the subgrid sale viscosity:

$$\mu_t = \rho\Delta^2 S_w \; (3)$$

The deformation parameter S_w is defined as:

$$S_w = \frac{S_d{:}S_d^{3/2}}{S_d{:}S_d^{5/4} + S{:}S^{5/2}} \; (4)$$

The tensor S_d is defined as:

$$S_d = \frac{1}{2}\left[\nabla\nu\cdot\nabla\nu + (\nabla\nu\cdot\nabla\nu)^T\right] - \frac{1}{3}tr(\nabla\nu\cdot\nabla\nu)I \;\; (5)$$

Where I is the identity tensor.

3. Results

CFD simulations were performed to numerically analyze the lower plenum of the HTTF. The velocity distributions within a cross-sectional plane at 50% height were presented in Figures 3 and 4. In Figure 3, the x-velocity contours are depicted, with (a) representing the results from the Reynolds-Averaged RANS model and (b) representing the results from the LES model. As shown by the x-velocity contours, flow separation, and vortex shedding were observed behind the column. In Figure 4, (a) presents the results from the RANS model, while (b) displays the results from the LES model. As illustrated by the y-velocity contours, it can be observed that the y-velocity on the side further from the duct is primarily influenced by the jet inlet. This results in higher velocities in the downward direction closer to the floor, with velocities decreasing as one approaches the exit.

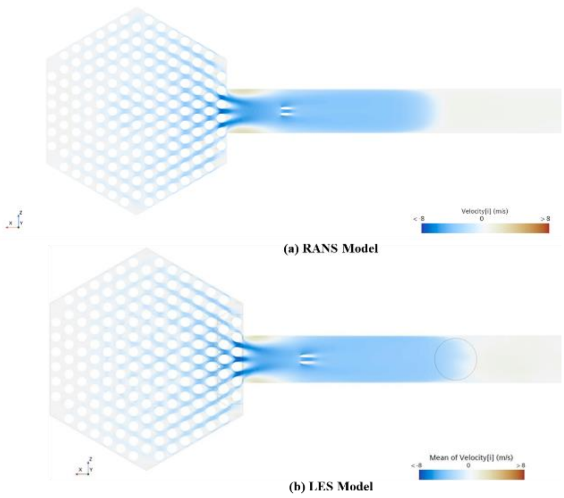

Fig. 3. Comparison results of x-velocity distributions between the RANS and LES models

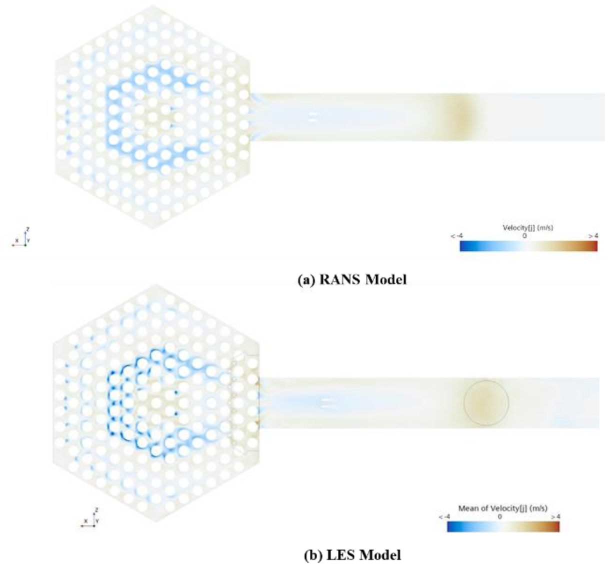

Fig. 4. Comparison results of y-velocity distributions between the RANS and LES models

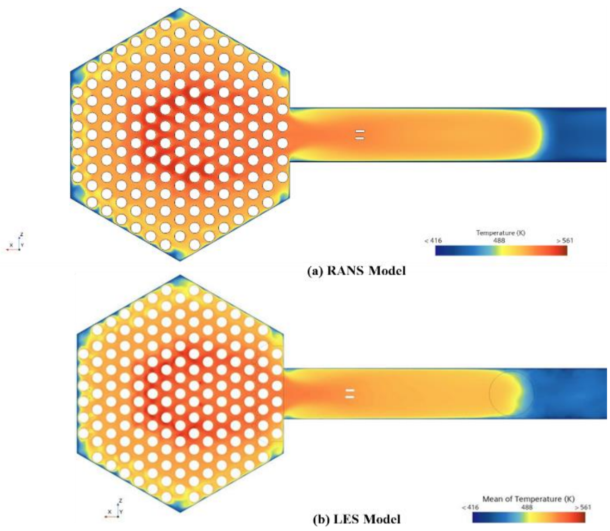

Fig. 5. Comparison results of temperature distributions between the RANS and LES models

As evident from the x-velocity and y-velocity contours, it can be observed that the LES model distinctly captures the eddy formations near the walls due to the boundary layer effects. This phenomenon was more pronounced in the LES model compared to the RANS model.

The fluid's temperature variation can be investigated through the analysis of HTTF's flow characteristics. The calculated temperature distribution is presented in Fig. 5. In Fig. 5, (a) represents the results from the RANS model, while (b) illustrates the results from the LES model. The temperature distribution pattern exhibited in Fig. 5 is similar to the velocity distribution trends observed in Fig. 3 and Fig. 4. Within Fig. 5, the temperature profiles demonstrate that the average temperatures at the edges of each side are relatively lower than the central temperature. This phenomenon indicates that along the edges, there exists a reduced temperature due to enhanced mixing and heat transfer. The presence of these lower temperatures at the edges can lead to the development of a higher temperature gradient in the direction perpendicular to the jet, resulting in heat conduction across those regions.

The analysis of flow characteristics revealed that there were subtle differences in velocity and temperature values between the inlet and outlet regions based on the turbulence modeling approach. However, the flow features resulting from turbulent eddy formation, and subsequently the convective mixing, were more distinctly captured by the LES model. In connection with this, when examining the influence on temperature distribution, the LES model excelled in more accurately predicting heat transfer and mixing processes within the fluid due to its more pronounced representation of small-scale turbulent structures. As a consequence, heat transfer around the rod and temperature mixing were better simulated, leading to a more accurate modeling of temperature distribution.

In contrast, the RANS model, due to its averaged representation of turbulent structures, provided comparatively modest predictions of small-scale mixing, potentially leading to relatively less accurate temperature distribution predictions. Moreover, changes in the intensity of turbulence and eddies directly affect energy transfer and mixing processes within the fluid. Notably, variations in the flow structure and intensity around the rod indirectly regulate heat transfer and consequently influence temperature distribution fluctuations. The enhanced flow structures and intensity changes, particularly in the vicinity of the rod, have an indirect yet significant impact on regulating heat transfer and driving temperature distribution variations.

4. Conclusions

In this study, computational fluid analysis of the flow characteristics in the HTTF experimental apparatus was conducted using Star-CCM+. Both LES and RANS models were employed to perform numerical simulations for the HTTF, and a comparative analysis was carried out. The RANS model, which utilizes time-averaging techniques, was compared with the LES model to assess their respective capabilities in capturing the flow behavior. The RANS model, based on time-averaging, exhibited the ability to reasonably replicate the overall flow characteristics compared to the LES model. However, significant discrepancies were observed in localized regions, as well as in terms of turbulent flow quantities and viscosity. These differences highlighted the limitations of RANS when simulating complex turbulent flows. Consequently, for a thorough analysis of the vortex behavior within the experimental apparatus, the LES model proved to be more effective. LES divides the energy of turbulence into different scales, directly resolving the large eddies with significant energy, while estimating the behavior of small eddies with lower energy using turbulence models. Consequently, in contrast to RANS models that represent turbulence energy with a single turbulence term, LES provides a more precise representation of the structure and motion of turbulence. Unlike RANS, LES accurately depicts the flow in the lower plenum, energy transfer, and other related phenomena, enhancing our understanding of turbulence. Furthermore, LES differs from RANS by directly simulating the structure and motion of turbulence, significantly reducing the need for subgrid scale models. This reduction leads to less uncertainty associated with model parameter adjustments. Therefore, LES offers a more detailed and accurate description of turbulence behavior and its impact on various flow characteristics, making it a valuable tool for improving our understanding of turbulent flows. The utilization of the LES model in this study yielded more accurate insights into the intricate turbulence patterns and local phenomena present in the HTTF experimental device. This underscores the superiority of the LES approach when studying complex turbulent flow scenarios, providing valuable information for further research and practical applications.

Acknowledgment

Institute of Energy Technology Evaluation and Planning(KETEP) grant funded by the Korea government(MOTIE)(RS-2023-00243201, Global Talent Development project for Advanced SMR Core Computational Analysis Technology Development)

This work was supported by the National Research Foundation of Korea(NRF) grant funded by the Korea government(MSIT) (No. 2021M2E2A2081062)

REFERENCES

[1] McEligot, D. M., and McCreery, G. E., "Fundamental thermal fluid physics of high temperature flows in advanced reactor systems", Idaho National Laboratory, Idaho Falls, ID, 2002

[2] B. G. Woods, “OSU High Temperature Test Facility Technical Design Report.”, OSU-HTTF-TECH-003-R1, Rev. 1, Oregon State University, 2017

[3] Stone & Webster Engineering Corp. “Preliminary Safety Information Document for the Standard MHTGR.” HTGR-86-024, Department of Energy ,1986

[4] Smagorinsky, Joseph. "General circulation experiments with the primitive equations: I. The basic experiment." Monthly weather review 91.3, pp 99-164, 1963

[5] FERZIGER, Joel H.; PERIĆ, Milovan; STREET, Robert L. Computational methods for fluid dynamics. springer, 2019.

[6] Siemens Digital Industries Software. Simcenter Star-CCM+ User Guide, version 2022.1. Siemens, 2022.