CFD Analysis of Turbulent Flow in Nose Piece for SFR Fuel Assembly

Yeong Shin Jeong [a,b], Wang Kee In [a*], Kyoung Ho Yoon[a], Jin Sik Cheon [a]
[a]*Korea Atomic Energy Research Institute, 989-111 Daedeok-daero, Yuseong-Gu, Daejeon 34057*
[b]*Ulsan National Institute of Science and Technology, 689-798 50 UNIST-gil, Ulju-gun, Ulsan*
[*]*Corresponding author: wkin@kaeri.re.kr*

1. Introduction

The fuel assembly for sodium-cooled fast reactor(SFR) consists of nose piece, lower/upper reflector, fuel bundle and lifting lug. The nose piece is an inlet nozzle with nine(9) side orifices. The coolant from core inlet plenum is forced to flow inside the nose piece through side orifices and directed upward to fuel bundle. Hence, the coolant flow in nose piece is highly unsteady turbulent flow and complex flow due to staggered side orifices. This paper presents the CFD(computational fluid dynamics) predictions of turbulent flow in nose piece and pressure loss coefficient of side orifices for SFR fuel assembly.

2. CFD Model and Method

A CFD model is created to simulate the coolant flow in SFR fuel assembly[1]. Since this study is focused on turbulent flow in nose piece, the CFD model simulates the side orifices in nose piece but neglected other fuel assembly components. Fig. 1 illustrates the SFR fuel assembly, nose piece and CFD model. The side orifices are staggered in three axial and azimuthal directions[2]. The CFD model consists of inlet chamber, nose piece with 9 side orifices and outlet chamber.

The hexahedral mesh is generated separately for inlet chamber, side orifice and outlet chamber as shown in Fig. 2. The three components are then connected by general grid interface(GGI) option. Total number of mesh is 3.2 million, 6.5 million and 9.7 million cells.

A constant velocity of coolant flow is given at inlet boundary and constant pressure is applied at outlet boundary. No-slip condition is also used at wall boundaries.

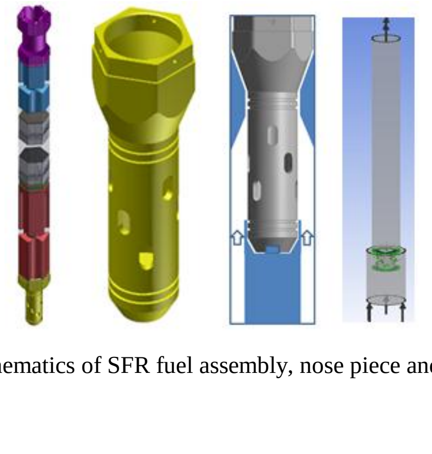

Fig. 1. Schematics of SFR fuel assembly, nose piece and CFD model.

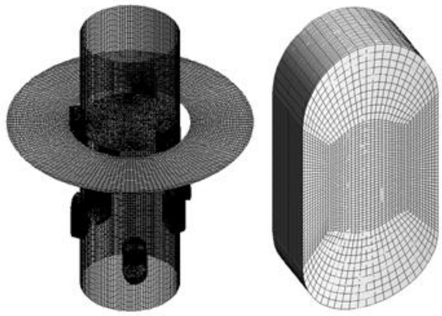

Fig. 2. CFD mesh for nose piece and side orifice(3.2M cells)

The CFD calculation was performed for the Reynolds number in side orifice (Re_o) of 10^4, 10^5 and $2x10^5$. Since the coolant flow in nose piece is unsteady and highly turbulent, a unsteady RANS simulation and large eddy simulation(LES) were conducted in this study. For the RANS simulation, this study used the standard k-ε model, SST-SAS model and SSG Reynolds stress model. The SST-SAS model is a modified version of SST model by the additional SAS(scale-adaptive simulation) source term in the transport equation for the turbulence eddy frequency. The constant Smagorinsky model is also used in this LES. The convergence criteria for this CFD solution is the RMS residual of governing equations being smaller than 0.0001. A commercial CFD code, ANSYS CFX 15.0[3] was used in this CFD analysis.

3. Results and Discussions

The CFD simulation predicted an unsteady three-dimensional turbulent flow in nose piece of SFR fuel assembly. Fig. 3 shows the axial velocity contour and vector in central vertical plane of nose piece using SSG Reynolds stress model for the orifice Reynolds number of 10^5. The CFD predictions show a violent swirling flow inside nose piece due to staggered array of side orifices in longitudinal direction. A recirculating flow seems to also appear in nose piece due to lateral flow through the three orifices in azimuthal direction. It is also noted that the flow distribution is non-uniform at the outlet of nose piece, i.e., an expanded flow region connected to the lower reflector of fuel assembly.

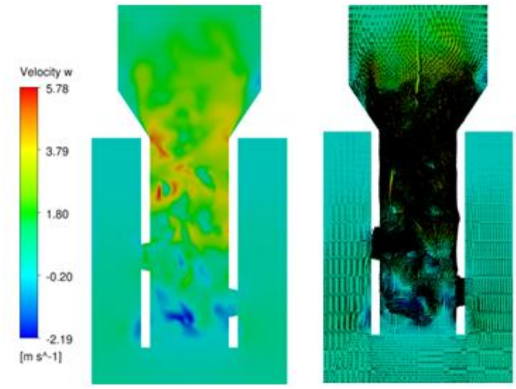

Fig. 3. Velocity contour and vector in nose piece for $Re_o=10^5$ using SSG Reynolds stress model.

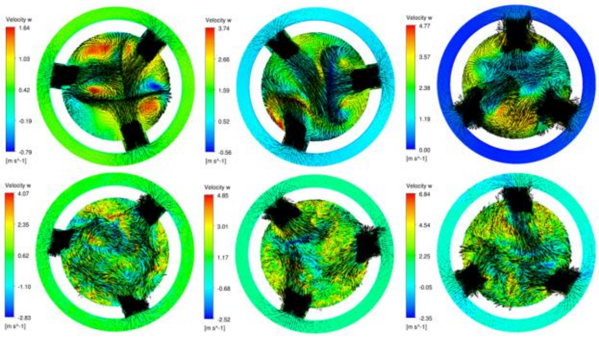

Fig. 4. Velocity contour and vector in side orifices at bottom, middle and top positions (from left) for $Re_o=10^5$ using SSG Reynolds stress model (top) and LES (bottom).

Fig. 4 shows the CFD predictions of axial velocity contour and vector in the horizontal plane at three axial locations of side orifices. From left, it shows the flow pattern at bottom, middle and top orifices. The side orifice appears to generate a high-speed jet flow inside the nose piece. The lateral flow from three orifices impinges each other in the central region of nose piece and creates a recirculating flow. The combination of lateral and axial flows generates a complex unsteady three-dimensional flow in the nose piece. The SSG Reynolds stress model seems to predict a stronger recirculating flow than the LES. However, the LES predicted a much larger variation of axial velocity than the SSG model. For instance, the axial velocity (w) for the middle orifice varies from -0.56 to 3.74 m/s and -2.52 to 4.85 m/s for the SSG model and the LES, respectively.

Fig. 5 shows the vorticity contour in nose piece predicted by the LES and SSG model. The vorticity here is defined as a magnitude by root-sum-squaring the vorticity components in three directions. It is noted that higher vorticity occurs in orifice region due to a high flow velocity. The LES predicted the vorticity contour significantly higher than the SSG model. The LES prediction shows a higher vorticity not only in the orifice but also in the inner region of nose piece.

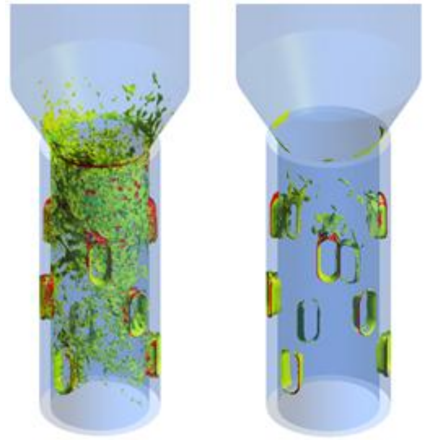

Fig. 5. Vorticity in nose piece for $Re_o=10^5$ using LES (left) and SSG Reynolds stress model (right).

A pressure loss coefficient of side orifice was estimated from the CFD simulation. The experiment[2] and the Idelchik correlation[4] give the orifice loss coefficient of approximately 3.5 and 3.7, respectively for $Re_o=10^5$. The CFD calculation predicts the loss coefficient of 2.4-3.1 depending on turbulence model. Table I shows a comparison of the pressure loss coefficient of side orifice estimated by CFD simulation, obtained from experiment results and Idelchik correlation (diagram 3-16) for $Re_o=10^5$ according to turbulence model.

Table. I: Comparison of pressure loss coefficient of side orifice for $Re_o=10^5$

		Value
Experiment[2]		3.5
Idelchik[4]		3.7
CFD	Standard k-ε	3.1
	SST-SAS	2.9
	SSG	2.8
	LES	2.4

4. Conclusions

A CFD analysis was performed to simulate coolant flow in nose piece of SFR fuel assembly. The unsteady CFD simulation was conducted using RANS turbulence models and LES. The CFD prediction shows a violent swirling and recirculating flow inside nose piece due to staggered arrangement of side orifices. The lateral and axial flows generate a complex unsteady three-dimensional flow in the nose piece. A higher vorticity was predicted to occur inside the nose piece particularly by the LES. The loss coefficient of the side orifice is estimated to be 2.5-3.0 which is somewhat lower than the experimental value of 3.5.

Acknowledgement

This work was supported by the National Research Foundation of Korea (NRF) funded by the Korea government (MSIP) (NRF-2012M2A8A2025646).

REFERENCES

[1] SFR development agency(KAERI), System Description of SFR Prototype Reactor, Daejeon, Korea, 2015.
[2] H. Y. Nam, J. M. Kim, K. W. Seo, and S. K. Choi, Development of an Experimental Correlation for a Pressure Loss at a Side Orifice, J. of Fluids Engineering, Trans. ASME, Vol.127, p. 388, 2005.
[3] ANSYS Inc., User Guide for CFX 15.0, 2013.
[4] I. E. Idelchik, Handbook of Hydraulic Resistance – diagram 3-16 (p. 136), 2nd edition, Hemisphere Publishing Corporation, New York, 1986.