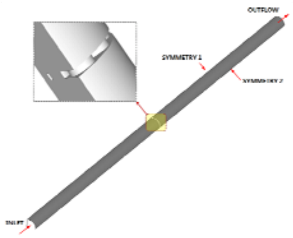
Analysis on a Fuel Assembly Simulator in the Reactor Flow Distribution Test for SMART

Y.I. Kim[a*], Y.M. Jeon[a], J. Yoon[a], H. Bae[a], Y.J. Chung[a], W.J. Lee[a]
[a]*Korea Atomic Energy Research Institute, 1045 Daedeok Street, Yuseong-gu, Daejeon 3050353, Korea*
[*]*Corresponding author: yikim3@kaeri.re.kr*

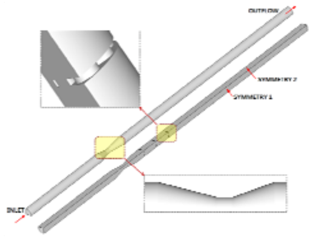
1. Introduction

CFD (Computational Fluid Dynamics) is being used in many problems where three dimensional (3D) behaviors are important. The flow distribution at the core inlet is one of the significant 3D phenomena to be figured out in a reactor [1]. A reactor flow distribution test has been being performed using a 1/20 Re and 1/5 size model for SMART. As the fuel assemblies of SMART are composed of many rods (17x17), the assemblies are imitated using simplified simulators in the test [2, 3].

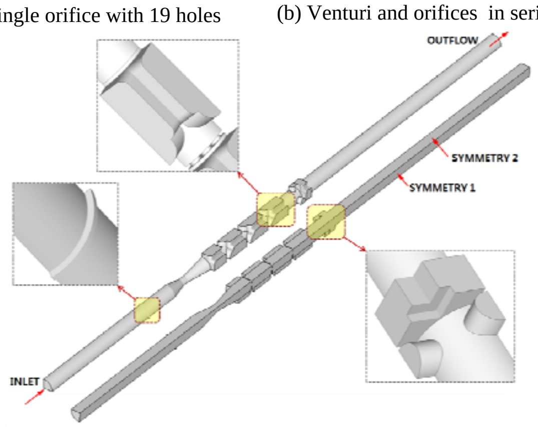
Meanwhile CFD analyses on the core inlet flow distribution will be performed in SMART. In CFD simulations since the local accuracy of a single simulator greatly influences on the flow distribution in the core, the local flow should be simulated accurately. As groundwork for the application in the core CFD analysis, a numerical analysis for the single simulator is performed in this paper. The accuracy of turbulence models and grid effect are investigated in this paper.



(a) Single orifice with 19 holes



(b) Venturi and orifices in series



(c) Fuel assembly simulator

Fig. 1. Configuation of computaional domain

2. Methods

2.1 Model description

With the assumption that flow is in steady state and fluid properties are constant, this simulation is carried out using single precision solver, and the SIMPLE algorithm for pressure velocity coupling, the 2nd-order upwind method for discretization, and standard wall function for RKE and RNG (the option of low Reynolds correction is not used for SST).

A commercial CFD code, Fluent 12 [3] is used in this paper. The governing equation for the three dimensional, incompressible, steady state, Newtonian, and turbulence flow is as follow;

$$\frac{\partial \langle u_i \rangle}{\partial x_i} = 0 \tag{1}$$

$$\rho \frac{\partial \langle u_i \rangle \langle u_j \rangle}{\partial x_j} = -\frac{\partial \langle p \rangle}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu \left(\frac{\partial \langle u_i \rangle}{\partial x_j} + \frac{\partial \langle u_j \rangle}{\partial x_i} \right) - \rho \langle u_i' u_j' \rangle \right] \tag{2}$$

Turbulence modes applied in this paper such as Shear Stress Transport (SST) k-ω, Renormalization Group (RNG) k-ε, Realizable k-ε (RKE) are well summarized in reference [4].

2.2 Configurations and Boundary Conditions

The fuel assembly is modeled in the simulator using a venturi, three orifices with 19 holes each installed in series [2, 3]. And the simulator has side holes to reproduce the cross flow effect in fuel assemblies (Fig. 1c).

For a comparison of CFD simulations to empirical correlations [5], the 3D-1/4 axisymmetric models for single orifice with several holes and for venturi and orifices combined in series are used as shown in Fig. 1a and 1b. And finally CFD analyses for a quarter model of the fuel assembly simulator are carried out (Fig. 1c). The boundary and operation conditions applied in the models are displayed in Fig.1 and references [2, 3].

3. Results and Discussion

3.1 Orifice composed with several holes

For the orifice with single hole, the CFD results using SST, RNG, and RKE are very close to empirical correlations within 10% deviation in reference [6].

The calculation results for the single orifice with 19 holes (Fig. 1a) are summarized in Table 1. As shown in Table 1, the deviation becomes large compared to the reference for the single hole [6]. To increase in deviation can be explained that the conditions of each hole in the orifice with several holes, such as the entrance and exit condition, are considerably different with those with the single hole

Table 1. Loss coefficients for the orifice with 19 holes

Turbulence model	Mesh (million)	Loss coefficients based on inlet		
		Empirical	CFD	Deviation (%)
RKE	4.8	2.29	1.59	31
RNG	4.8	2.29	1.71	25
SST	4.8	2.29	1.82	20

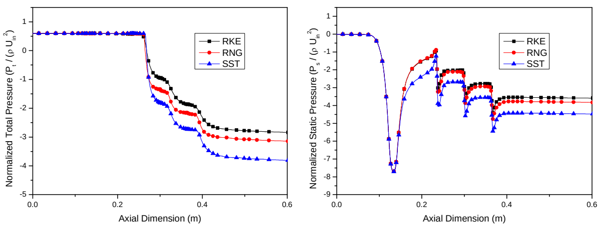
3.2 venturi and orifice in series

The calculation for venturi is performed in reference [7]. The CFD results are very close to empirical correlations [5] within 7% deviation in SST, but far from the correlations over approximate 30% in RNG and RKE. The loss coefficient of venturi is very small, so it is sensitive to the minor change in flow patterns.

The CFD results for the series (Fig. 1b) using SST, RNG, and RKE are clarified in Table 2. All turbulence models underestimate the loss coefficients over 10%. But in the empirical correlations the coupling effect between venturi and orifice or between orifice and orifice is not considered, so we can easily find that the correlations have some errors. The total pressure and static pressure at axis line of the series for RKE, RNG, and SST are showed in Fig. 2.

Table 2. Loss coefficients for venturi and orifices in series

Turbulence model	Mesh (million)	Loss coefficients based on inlet		
		Empirical	CFD	Deviation (%)
RKE	12.0	10.0	7.1	30
RNG	12.0	10.0	7.6	25
SST	12.0	10.0	8.9	11



(a) Total pressure (b) Static pressure

Fig. 2. Pressure variation with axis

3.3 Simulator of fuel assemblies

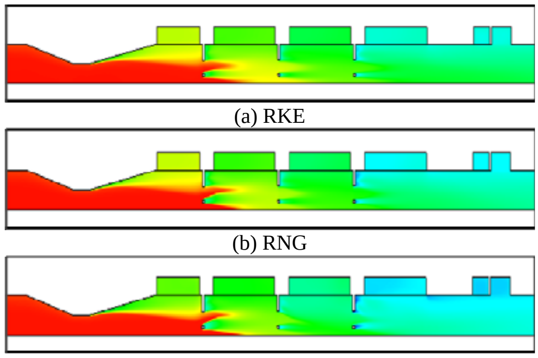
The simulation results for the simulator using SST, RNG, and RKE are displayed in Table 3 and Fig. 3. RKE makes an estimate of the test result [3] within 1% accuracy, RNG over 9% and SST over 25%.

As shown in Table 3, the deviation in SST remarkably increases compared to the separated CFD results in ref. [6, 7]. SST is highly overestimating the coupling effect between venturi and orifice or between orifice and orifice. In this case, the conditions of entrance and exit, compared to the fully developed condition, are notably altered.

Even though RKE has a weakness in the simulation for the venturi, RKE prediction for the calibrations test, where the overall loss coefficients in all single simulators are tested, are most accurate in all turbulence models investigated. Figure 3 shows the total pressure contours on one symmetric plane of the simulator for RKE, RNG, and SST.

Table 3. Loss coefficients for the fuel assembly simulator

Turbulence model	Mesh (million)	Loss coefficients based on inlet		
		Test [7]	CFD	Deviation (%)
RKE	0.83	8.40	8.37	0.3
RKE	1.15	8.40	8.41	0.1
RKE	12.50	8.40	8.35	0.5
RNG	12.50	8.40	9.13	8.7
SST	12.50	8.40	10.67	27.1



(a) RKE

(b) RNG

(c) SST

Fig. 3. Contours of total pressure

4. Conclusions

Numerical analyses using Fluent 12 are performed for the fuel assembly simulator applied in the reactor flow distribution test for SMART. In the single orifice with 19 holes, the loss coefficient decreases because of the coupling effect between holes. Also in the venturi and the orifices series, the loss coefficient decreases because of the change of in and out conditions. The realizable *k-ε* turbulence model predicts the calibrations test in the fuel assembly simulator more accurate. This model is recommended for the CFD simulation associated with the core flow.

Acknowledgement

This study has been performed under a contract with the Korean Ministry of Educational Science and Technology.

REFERENCES

[1] K.B. Lee, et.al., YGN 3&4 Reactor Flow Model Test, Journal of KNS, Vol.23, p.340, 1991
[2] H. Bae, et.al., Design Concept of a 1/5-Scale Core Simulator, Tans. of KNS spring meeting, 2010.
[3] D.J. Euh, et.al, Calibration Report for the SG and Core Simulator of the Reactor Flow Distribution Test for SMART, 752-TF-498-004, KAERI, 2010.
[4] Fluent 12.0 Manual, ANSYS Inc., 2009.
[5] I.E. Idelchik, Handbook of Hydraulic Resistance, third edition, Begell house, 2000.
[6] Y.I. Kim, SMART Steam Generator Orifice Size Calculation, KAERI/100-NH301-003/2010, KAERI, 2010.
[7] J. Yoon, et.al., Numerical Simulation on the Venturi in SMART Reactor Flow Distribution Test Facility, Tans. of KNS spring meeting, 2011.