A Numerical Study for the Effect of the Accuracy Order of the Discretization Scheme on the Prediction Performance for the Flow Distribution inside a Fuel Assembly with Twist-Split Type Mixing Vane

Gong Hee Lee [a,b*], Jin Sung Park [a], Ae Ju Cheong [a]
[a]*Nuclear Safety Research Department, Korea Institute of Nuclear Safety, Daejeon, 34142, Korea*
[b]*Nuclear and Radiation Safety Department, University of Science and Technology, Daejeon 34133, Korea*
[*]*Corresponding author: ghlee@kins.re.kr*

1. Introduction

Spatial discretization errors result from both the numerical order of accuracy of the discretization scheme and grid spacing. It is well known that second, or higher, order discretization schemes are potentially able to produce high-quality solutions. In addition, when the flow either is not aligned with the grid or is complex, it is recommended that the first order discretization scheme not be used for the convection term, if possible [1]. However, the higher-order scheme can also result in convergence difficulties and instabilities at certain flow conditions.

In this study, to examine the effect of the numerical order of accuracy of the discretization scheme on the prediction performance for the flow distribution inside a fuel assembly with twist-split type mixing vane, simulations were conducted with the commercial CFD (Computational Fluid Dynamics) software, ANSYS CFX R18.1 [2]. The predicted results were compared with the measured data.

2. Analysis Model

As shown in Fig. 1, OFEL (Omni Flow Experimental Loop) test facility [3] was used to measure the flow distribution inside a fuel assembly with twist-split type mixing vane. Test rig consisted mainly of a water storage tank, a centrifugal pump, a flow meter, and a test section. Test section was made up of a square housing, fixing plate, rod bundle, rod support, and spacer grid with twist-split type mixing vane. Length and outer diameter (D) of a fuel rod, and rod-to-rod pitch (P) were 2,000 mm, 25.4 mm, and 34.29 mm, respectively. Therefore, P/D was 1.35 which corresponded to the regular rod-bundle configuration.

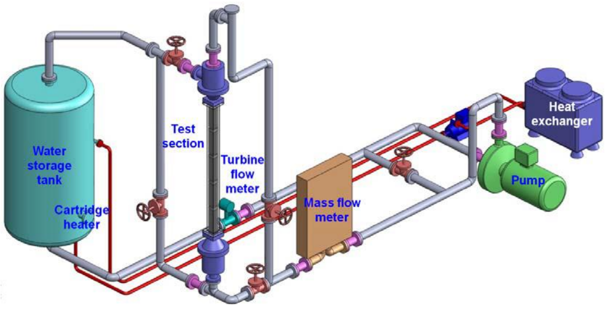


Fig. 1. Schematic diagram of OFEL facility [3].

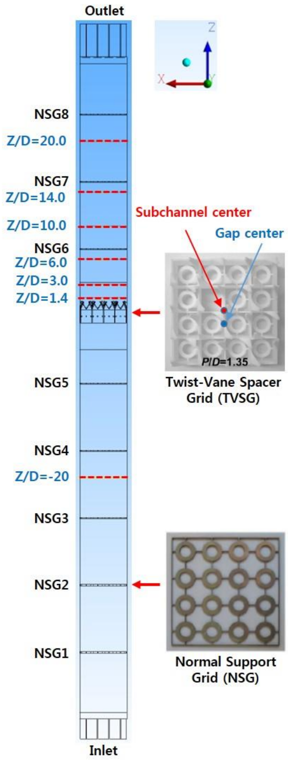


Fig. 2. Schematic diagram of test section and measurement positions for velocity components.

As shown Fig. 2, velocity distribution inside the subchannel was measured at several cross-sectional planes by using either PIV (Particle Image Velocimetry) or LDV (Laser Doppler Velocimetry).

Inlet temperature and pressure condition of the working fluid were measured at 35℃ and 0.1 MPa, respectively. The bundle-average axial velocity was estimated to be 1.5 m/s.

3. Numerical Modeling

The flow inside the fuel assembly was assumed to be steady, incompressible, isothermal and turbulent. Two different types of the discretization scheme for the convection-terms-of-momentum and -turbulence equations, i.e. 1st order upwind scheme and a high resolution scheme, were used.

The solution was considered to be 'converged' when the residuals of variables were below 10^{-5} and the variations of the target variables were small. Simulation was conducted with the commercial CFD software, ANSYS CFX R18.1.

The Shear Stress Transport (SST) model was used to simulate the turbulent flow inside a fuel assembly. This model may give highly accurate predictions of the onset and the amount of flow separation under adverse pressure gradients. It is also recommended for accurate boundary layer analysis.

Fig. 3 shows the grid system for the computational domain that had the same size as the test facility. A hybrid mesh, made up of tetrahedrons, wedges, pyramids and hexahedrons, was generated to prevent the oversimplification of the geometry, and to have more efficient mesh distribution. Prism layers were used to get higher resolution in the near-wall region. Total numbers of elements were 3.4×10^7.

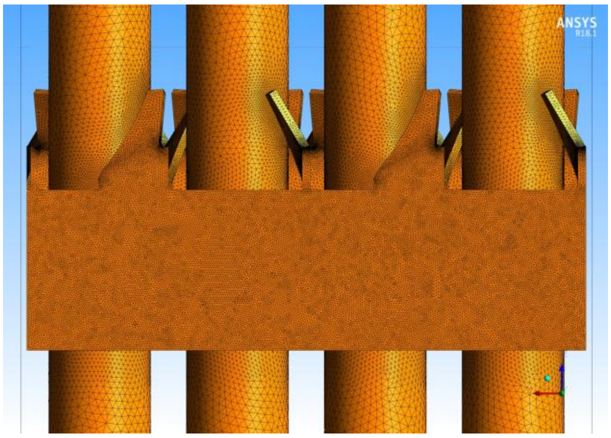


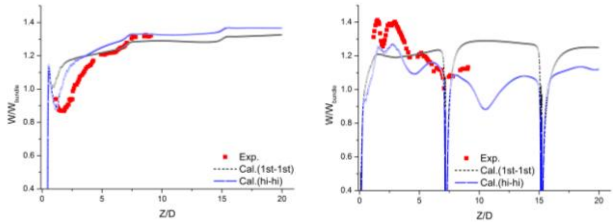
Fig. 3. Grid system.

Because the flow area at the inlet of test section is different from that in the flow passage, uniform velocity, calculated by considering the above-mentioned condition, was used as an inlet-boundary condition. The 'average pressure over the whole outlet' option; with a relative pressure of 0 Pa, was used as an outlet-boundary condition. A no-slip condition was applied at the solid wall. To model the flow in the near-wall region, the automatic near wall treatment method was applied.

4. Results and Discussion

Fig. 4 shows the comparison of axial velocity profiles, measured and calculated along the z-axis (downstream of mixing vane) at both the subchannel center and the gap center of rod-to-rod. Both positions were indicated by the solid circles in Fig. 2.

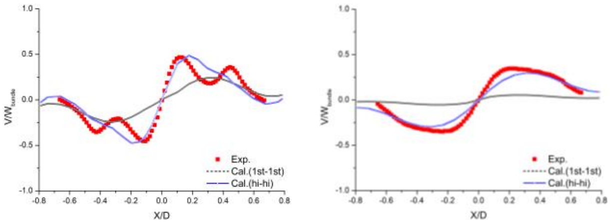
As shown in Fig. 4(a), the 1st order accurate upwind differencing (1st-1st) over-predicted minimum axial velocity near the mixing vane tip in the subchannel center; compared with high resolution scheme (hi-hi). Additionally, at the gap center of rod-to-rod, the 1st order accurate upwind differencing (1st-1st) could not predict the gradual decrease of axial velocity in the downstream region (Z/D~7.5) of the mixing vane. (see Fig. 4(b))



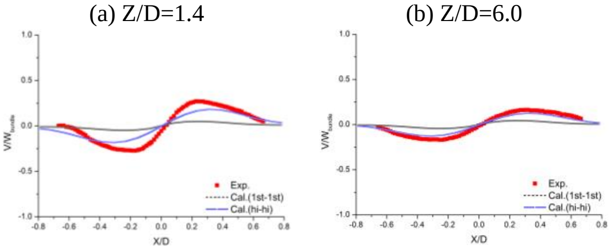
(a) subchannel center (b) gap center between rods

Fig. 4. Streamwise variation of axial velocity profile in the central subchannel.

Fig. 5 shows the lateral velocity (y-axis velocity component) distribution along a horizontal line in the central subchannel. The 1st order accurate upwind differencing (1st-1st) under-predicted peak velocity magnitude and resulted in less variation in the velocity magnitude; compared with high resolution scheme (hi-hi). The reason is that the 1st order accurate upwind differencing may increase the numerical/false diffusion and may not provide the desired accuracy. In the other hand, the computational results with high resolution scheme showed generally similar velocity profiles in comparison with the measured data.



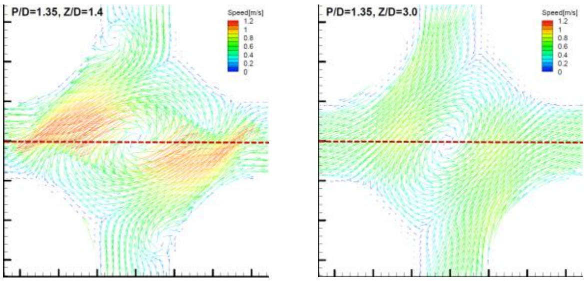
(a) Z/D=1.4 (b) Z/D=6.0



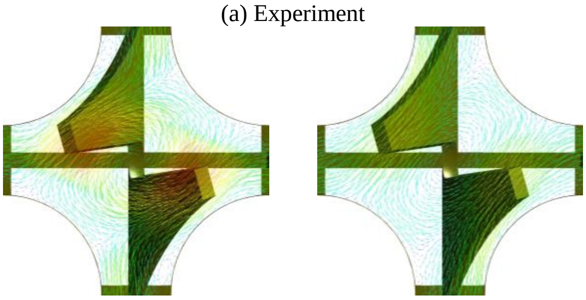
(c) Z/D=14.0 (d) Z/D=20.0

Fig. 5. Lateral velocity profile along the centerline in the central subchannel per the selected cross-sectional planes

Fig. 6 shows the comparison of velocity vector in the central subchannel. At the location Z/D = 1.4, close to the mixing vane, a large elliptic vortex was generated in the subchannel center and two small secondary vortex appeared in the peripheral region near the upper and lower gaps between rods. While the computational results with high resolution scheme (hi-hi) could capture well all of these vortex structures, the 1st order accurate upwind differencing (1st-1st) could not represent two small secondary vortex in the peripheral region near the upper and lower gaps between rods.



(a) Experiment


(b) Computation (hi-hi)

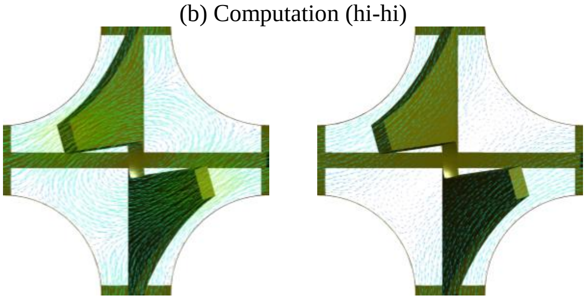

(c) Computation (up-up)

Fig. 6. Velocity vector in the central subchannel per the selected cross-sectional planes (left: Z/D=1.4, right: Z/D=3.0)

At the location Z/D = 3.0, the computational results with the 1st order accurate upwind differencing (1st-1st) showed the significantly attenuated swirl flow in the subchannel center, which is quite different from the experimental results.

5. Conclusions

In this study, to examine the effect of the numerical order of accuracy of the discretization scheme on the prediction accuracy for the flow distribution inside a fuel assembly with the twist-split type mixing vanes, simulations were conducted with the commercial CFD software, ANSYS CFX R18.1. The predicted results were compared with the measured data from OFEL facility. Through these comparisons, it was concluded that high order accurate schemes for the convection terms could guarantee the improved prediction accuracy to some extent. Similar conclusion was obtained from benchmark simulation for the MATiS-H test facility [4].

Whether mesh independent CFD solution can finally be obtained on the affordable mesh resolution is the most significant question to answer for reactor safety problems. If licensing applicants use CFD solution for their licensing documents, they should preferentially try to demonstrate that the final result of the calculations is mesh independent. Additionally, if the above-mentioned requirement is not satisfied, they should conduct the sensitivity study for the numerical order of accuracy of the discretization scheme, and use the physically appropriate and conservative simulation results for their licensing documents.

ACKNOWLEDGEMENT

This work was supported by the Nuclear Safety Research Program through the Korea Foundation Of Nuclear Safety (KOFONS) using the financial resource granted by the Nuclear Safety and Security Commission (NSSC) of the Republic of Korea (No. 1305002 & 1805007). The authors gratefully thank Dr. In, Wang-Kee in the KAERI for providing the OFEL experimental data and giving the valuable technical comments.

REFERENCES

[1] F. Menter, CFD Best Practice Guidelines for CFD Code Validation for Reactor Safety Applications, ECORA CONTRACT N° FIKS-CT-2001-00154, 2001.
[2] ANSYS Inc., ANSYS CFX, Release 18.1.
[3] C.-H. Shin, C. Lee, C.-Y. Lee, and W.-K., In, IAEA Benchmark Specification and Experimental Data: Flow Mixing in a 4x4 Rod Bundle with a Mixing-Vane Grid, LWR Fuel Development Division, KAERI, 2015.
[4] G. H. Lee and A. J. Cheong, Numerical Analysis of Flow Distribution inside a Fuel Assembly with Split-type Mixing Vanes for the Development of Regulatory Guideline on the Applicability of CFD Software, Korean Journal of Air-Conditioning and Refrigeration Engineering, Vol. 29, p. 538, 2017.