Transactions of the Korean Nuclear Society Autumn Meeting
Gyeongju, Korea, October 24-25, 2013

Computational Fluid Dynamic Simulation of Square Array Rod Subchannel

Hyung M. Son[a], Kune Y. Suh[a,b*]
[a]*Seoul National University, 1 Gwanak Ro, Gwanak Gu, Seoul 151-744, Korea*
[b]*PHILOSOPHIA, Inc., 1 Gwanak Ro, Gwanak Gu, Seoul 151-744, Korea*
[*]*Corresponding author: kysuh@snu.ac.kr, kysuh@4plusd.com*

1. Introduction

Crucial safety concerns are imposed upon fuel region where fission reactions take place. The rod array configuration is commonly utilized to maintain good heat transfer characteristics and the structural integrity. Design of the fuel assembly requires the various engineering considerations such as pressure drop, manufacturability, and generation of turbulence. A best way to estimate performance of the fuel assembly is to build a mockup and measure its physical parameters. However, the required cost and time render it rather difficult to perform experiments for all designs. Instead, with advances in numerical simulation techniques, computational fluid dynamics methods are extensively utilized to assist the experiments by reducing the trial and errors in the mockup. A numerical simulation is carried out on the bare square array rod geometry as a preliminary exercise for the grid spacer design. The results are compared against the data available in the literature to check on the simulation accuracy.

2. Problem Description

The geometry and the operating condition are first described of the test apparatus. Then the computational domain is discussed of the analysis geometry.

2.1 Experimental Apparatus

The experimental work carried out by Hooper [1] is selected for the numerical simulation. Figure 1 shows the cross section view of the test geometry where six aluminum circular tubes were held together using the spacer strips, creating two subchannels (yellow region) with a single open gap in-between. The rod diameter was 14 cm and the length was 9.14 m which gave enough room for flow development. Two test sections with differing pitch-to-diameter ratios (P/D= 1.107 and 1.194) were used. The smaller P/D was analyzed in this study. For the analysis on the larger P/D geometry Horváth and Dressel [2] carried out detailed numerical simulation using various turbulence models which ended up good agreement. In the test, air was used as working fluid and supplied to the section using a centrifugal blower. The measured data were presented for Reynolds number 48,400.

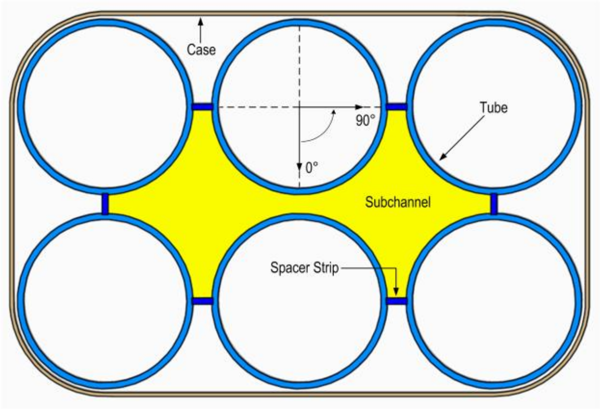

Fig. 1. Cross section view of test rig showing subchannels.

2.2 Computational Domain

In this calculation ANSYS CFX 12.1 software was used which solves the Navier-Stokes equation using the finite volume method [3]. Considering that the higher resolution of the mesh structure is usually accompanied by increased memory consumption and computational time, selection of the compact analysis domain is vital. The single subchannel on the right side is modeled using the symmetric boundary condition based on the symmetric configuration of neighboring subchannels, as shown in Fig. 2. The semi-fine mesh structure with a total of 9,809,902 hexahedral elements are used to simulate the flow with a reasonable accuracy where nondimensionalzed first wall mesh size (y^+) is kept around 15 based upon literature [4]. In order to make out the effect of wall boundary at the gap (top, bottom and right side), a 1/8 symmetric model is also analyzed.

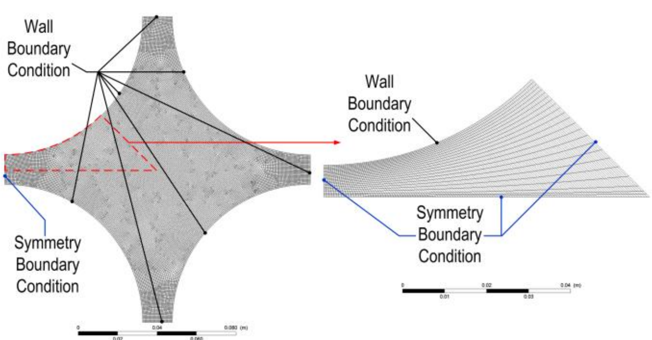

Fig. 2. Top view of discretized analysis domain (left: full subchannel, right: 1/8 subchannel).

3. Simulation Results

The Speziale-Sarkar-Gatski (SSG) Reynolds stress turbulence model is utilized to consider an anisotropic behavior of turbulence [5]. The standard k-ε turbulence model is also used for comparison. The calculations are continued until the root-mean-square (RMS) residuals of constituting equations fall under 1.0e-5. Figure 3 shows the radial velocity distribution at the outlet where the rotating cells are observed in near symmetric configuration. The full subchannel simulation reveals higher secondary flows (maximum: 2.503e-2 m/s vs. 1.626e-2 m/s) in those areas along the tube walls when compared against the 1/8 subchannel case on account of the existence of gap walls.

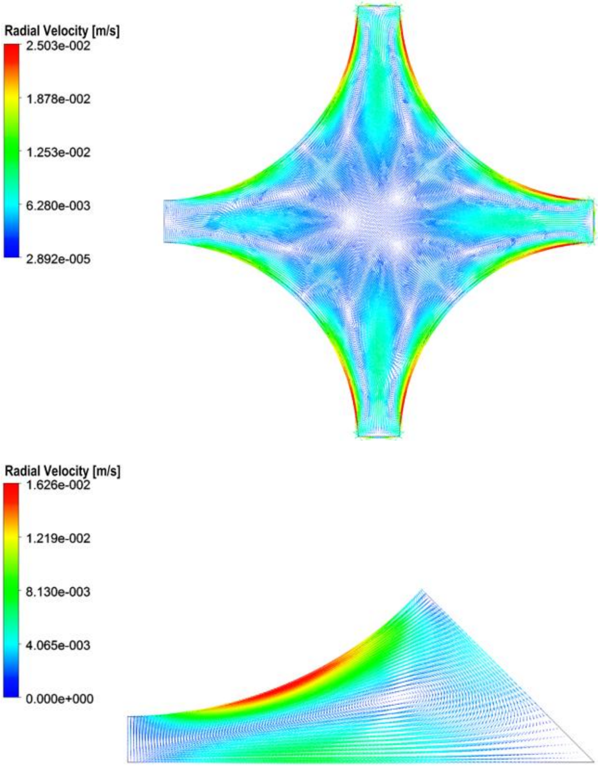

Fig. 3. Radial velocity distribution at outlet region (top: full subchannel, bottom: 1/8 subchannel).

Figure 4 compares the azimuthally measured (angle shown in Fig. 1) wall shear distribution from the experiment against the calculated values. The results from the 1/8 geometry does not capture the reduced shear stress in the gap wall region, showing symmetric distribution with respect to the 45° location. In addition, when compared against SSG, the k-ε turbulence model gives increased shear stresses at 45°. This might be related to the secondary flows [6]. Application of the SSG model gives more accurate results between 0° and 45° predicting a maximum before 45°. However, unlike what is measured in the experiment, the same model predicts two shear stress peaks between 0° and 90°, which is reported to exist in the higher P/D test case [1].

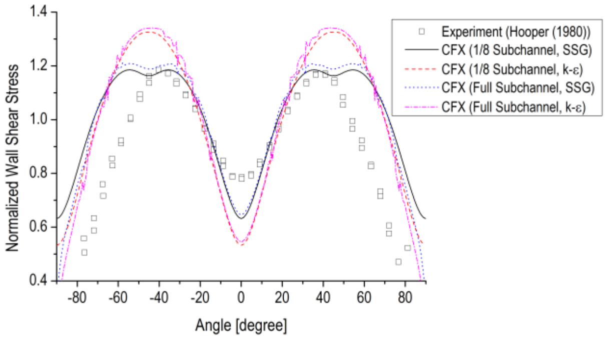

Fig. 4. Comparison of normalized wall shear distribution.

4. Conclusions

In the simulation of the subchannel geometry, appropriate selection of the boundary conditions in the gap region is important in obtaining good predictions. The Reynolds stress turbulence model shows improved calculation accuracy over the eddy diffusivity model by accounting for the effect of anisotropy in turbulence. The multiple shear stress peaks differ in the calculation from the test results calling for further investigation. However, the lack of other turbulence related data in the 45° to 90° in the same literature may necessitate a dedicated new set of experiment.

Acknowledgement

This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Korea government (MSIP) (No. 2008-0061900).

REFERENCES

[1] J.D. Hooper, Developed Single Phase Turbulent Flow Through A Square-Pitch Rod Cluster, Nuclear Engineering and Design, Vol. 60, p. 365, 1980.
[2] Á. Horváth, B. Dressel, Numerical Simulations of Square Arrayed Rod Bundles, Nuclear Engineering and Design, Vol. 247, p. 168, 2012.
[3] ANSYS, "ANSYS CFX-Solver Modeling Guide, Release 12.1," ANSYS, Inc., Canonsburg, PA, USA, 2009.
[4] X. Cheng, N.I. Tak, CFD Analysis of Thermal-Hydraulic Behavior of Heavy Liquid Metals in Sub-channels, Nuclear Engineering and Design, Vol. 236, p. 1874, 2006.
[5] C.G. Speziale, S. Sarkar, T.B. Gatski, Modelling Pressure-strain Correlation of Turbulence: An Invariant Dynamical Systems Approach, Journal of Fluid Mechanics, Vol. 227, p. 245, 1991.
[6] K. B. Lee, Analytical Prediction of Subchannel Friction Factor for Infinite Bare Rod Square and Triangular Arrays of Low Pitch to Diameter Ratio in Turbulent Flow, Nuclear Engineering and Design, Vol. 157, p. 197, 1995.