Comparison between CFD Analysis and Experimental Data for Flow in a 5 × 5 Rod Bundle with Spacer-Grids

Peng Yuan[1], Jin Yan[1], Yiban Xu[1], and Zeses Karoutas[1]

[1]*Westinghouse Electric Company, Cranberry Township, PA, USA*

yuanp@westinghouse.com

INTRODUCTION

Multi-physics virtual reactor models have been developed under sponsorship of DOE's Consortium for Advanced Simulation of Light Water Reactors (CASL) project [1]. This paper is to benchmark the CFD analysis with experimental data for the flow in a 5 × 5 rod bundle with spacer-grids. The Particle Image Velocimetry (PIV) experimental data was acquired at the Advanced Optical Multiphase Flow Research Laboratory (AOMRL) of Texas A&M University. The test rig includes a 5 × 5 rod bundle with four spacer grids and two vaneless support grids. The CFD model represents the geometry of the middle section of the test rig. Both steady state and transient CFD simulations were carried out. Axial velocity, lateral velocity and other variables from CFD simulations are compared with experiment data.

DESCRIPTION OF THE CFD MODELING

The tested rod bundle consists of 25 plastic rods with an OD of 9.5 mm placed in a 5 × 5 configuration. The rods are kept in place using four spacer-grids with Westinghouse V5H mixing vanes and two Westinghouse simple support grids. The locations of the six spacer-grids along the flow housing and its corresponding names are shown in Fig. 1.

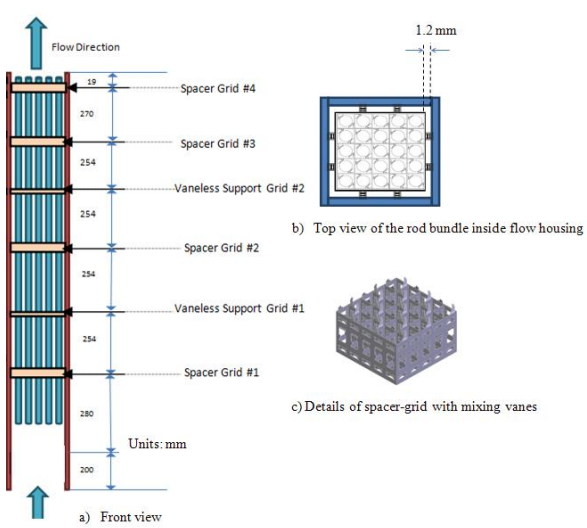

Fig. 1. 5 × 5 Rod bundle inside flow housing

The CFD simulation section includes one V5H spacer grid and one vaneless support grid. The simulation section starts from the middle of first V5H spacer grid and first support grid and ends at the middle of second spacer grid and second support grid, as shown in Fig. 2.

The distance from inlet to the bottom surface of support grid is 107.95 mm which is about the half of the fluid height between Spacer grid #1 and support grid #2. The distance from the top surface of support grid (not including vane) to outlet is also 107.95 mm, which is about half of the fluid height between Spacer grid #2 and support grid #3.

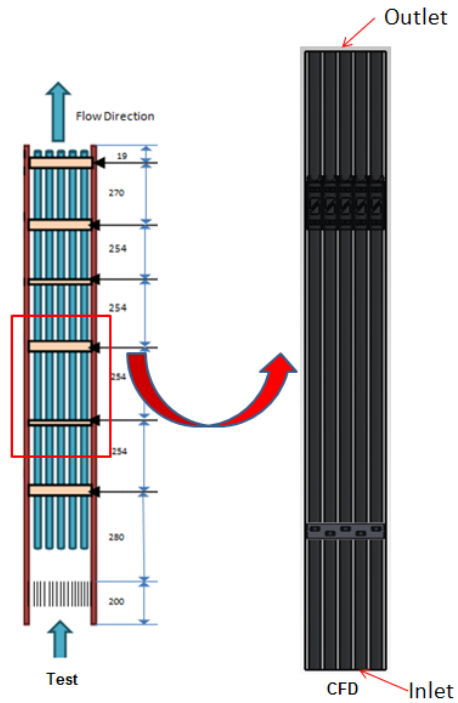

Fig. 2. CFD Simulation domain

Two types of simulations were carried out, i.e., steady state simulation and transient simulation. For the steady state simulation, several runs were carried out using different turbulence models with both STAR-CCM+ solver (version 7.02.008) [2] and FLUENT solver (version 14) [3]. Transient simulation was carried out using LES turbulence model with STAR-CCM+ solver only. The run matrix is listed in Table 1.

©2013 Westinghouse Electric Company LLC
All Rights Reserved

Table 1: Run Matrix

Run No.	Run Type	Solver	Mesh	Turbulence Model
1	Steady State	STAR CCM+	Mesh 1	Standard K-E
2	Steady State	STAR CCM+	Mesh 1	Realizable K-E
3	Steady State	STAR CCM+	Mesh 1	SST
4	Transient	STAR CCM+	Mesh 1	LES
5	Steady State	FLUENT	Mesh 2	Standard K-E
6	Steady State	FLUENT	Mesh 2	Realizable K-E
7	Steady State	FLUENT	Mesh 2	SST
8	Steady State	FLUENT	Mesh 2	RSM

Boundary Conditions and Fluid Properties:
Inlet: Velocity inlet of 2.48 m/s.
Outlet: Pressure outlet of zero relative pressure.
All other boundaries are no-slip wall boundary.
Fluid is water at 1 atm. and 24˚C. Water density is 997.561 kg/m^3 and water viscosity is 8.8871E-4 Pa-s.

RESULTS

Data sample locations

The data sample locations are shown in Fig. 3.

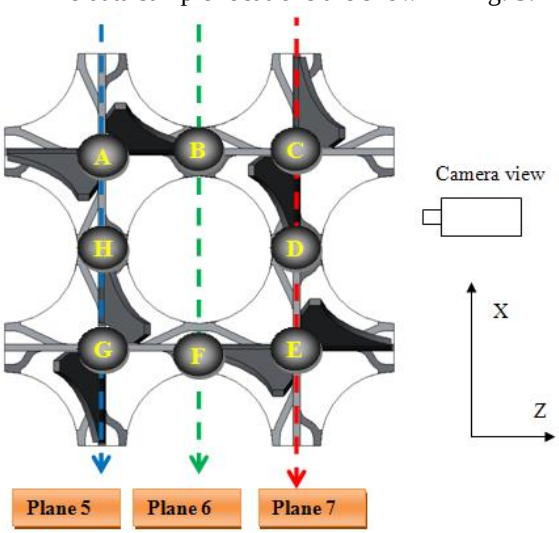

Fig. 3. The data sample locations (Viewing from Top, i.e., from downstream location)

Steady State Result Comparison

The comparisons were made between CFD simulation and experimental data for velocity U (axial velocity) and velocity V (lateral velocity) component on the 3 representative axial lines (line B, E and H) and on 3 planes (plane 5, 6 and 7). Result on one line and one plane were presented here

Fig. 4 shows the velocity comparisons at line B.

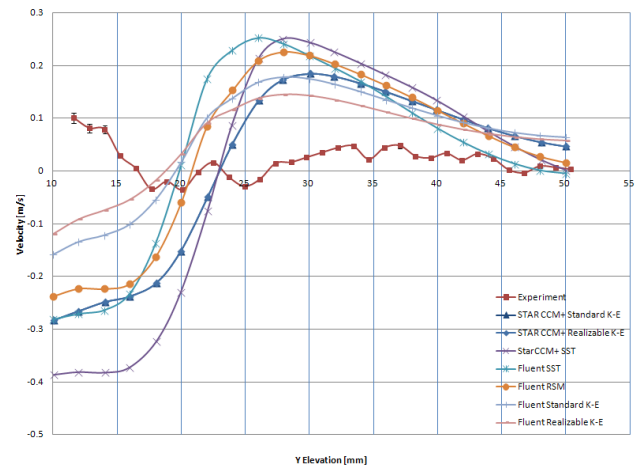

Fig. 4a: Lateral velocity at line B

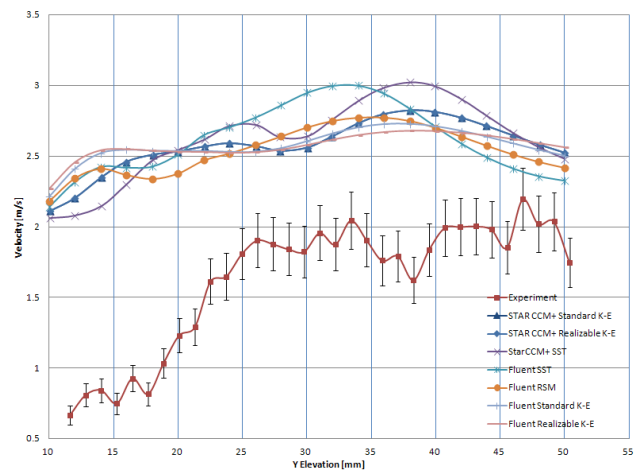

Fig. 4b: Axial velocity at line B

Fig. 5 shows the velocity contour plot comparison at Plane 5.

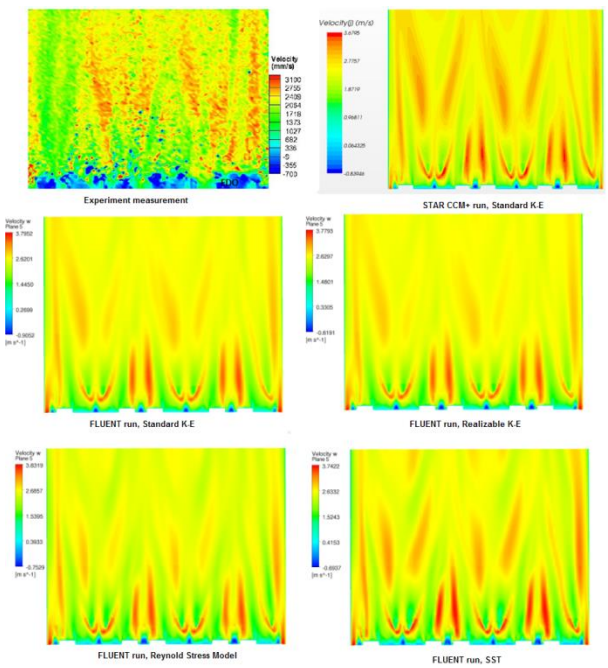

Fig. 5a: Comparison of axial velocity at plane 5

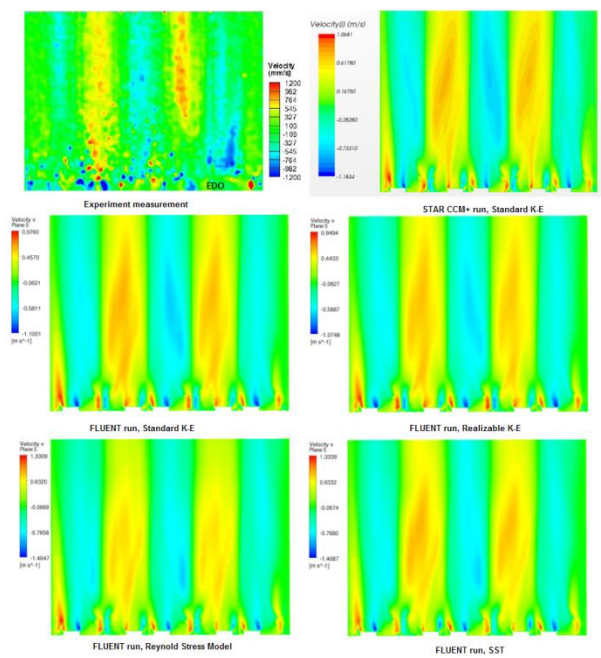

Fig. 5b: Comparison of lateral velocity at Plane 5

Transient Result Comparison

Transient simulation using Large Eddy Simulation (LES) turbulence model was carried out.

To compare the frequency of the CFD simulation and the experiment data, FFT (Fast Fourier Transform) was applied on both the experimental data and CFD result for the U and V velocities on points A through H. For the CFD result, FFT was applied on both the velocity at a point and the volume averaged velocity around a point. The comparison was shown in Fig. 6a and 6b for the representative Point D.

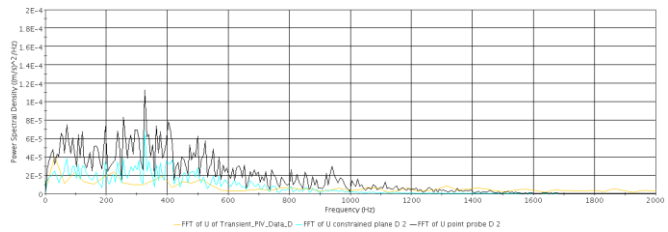

Fig. 6a: FFT of U velocity (axial velocity) at Point D

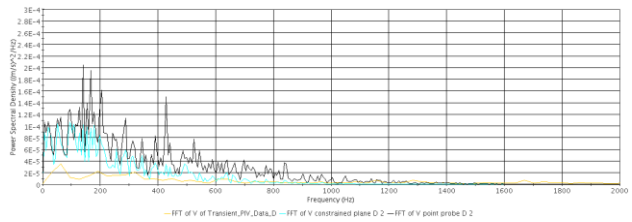

Fig. 6b: FFT of V velocity (lateral velocity) at Point D

It can be seen from Fig. 6a and 6b that the frequency calculated by CFD matches well with experiment data, especially for the volume averaged velocity data, meaning that the CFD simulation using LES model captured the inherent flow frequencies.

CONCLUSIONS

Using current state of the art experimental techniques, the experimental data was acquired at the Advanced Optical Multiphase Flow Research Laboratory (AOMRL) of Texas A&M University. The test rig includes a 5 × 5 rod bundle with four spacer grids and two simple support grids. The CFD model represents the geometry of the middle section of the test rig. Both steady state and transient simulation were carried out.

The comparison between CFD result and experiment data is shown. It can been seen from figures that for the steady state result, the CFD result is in relative good agreement with experiment data in terms of trend and absolute value. The agreement between CFD and experiment was also demonstrated by the resemblance between CFD result and experiment data for the contour plots of axial and lateral velocity component.

Transient simulation using Large Eddy Simulation (LES) turbulence model was also carried out. To compare the frequency of the CFD simulation and the experiment data, FFT was applied on both the experimental data and CFD result for the U and V velocities on points A through H. It can be seen from Fig 6a and 6b that the frequency calculated by CFD matches well with experiment data,

especially for the volume averaged velocity data, meaning that the CFD simulation using LES model captured the inherent flow frequencies.

ACKNOWLEDGEMENTS

The authors gratefully acknowledge the sponsorship of DOE's CASL project, with special thanks to Dr. Elvis Dominguez-Ontivero at Texas A&M University.

REFERENCES

1. Yan, J. et al., Multi-Physics Computational Models Development for Westinghouse PWRs, to be submitted to ANS 2013 Winter Meeting, November 2013, Washington DC.
2. CD-adapco STAR-CCM+ 7.02.008 User's Guide, www.cd-adapco.com
3. ANSYS Fluent 14 User's Guide, www.ansys.com