Performance Comparison of the Commercial CFD Software for the Prediction of Turbulent Flow through Tube Bundles

Gong Hee Lee*, Young Seok Bang*, Sweng Woong Woo*
*Korea Institute of Nuclear Safety, Daejon, Republic of Korea
*Corresponding author: ghlee@kins.re.kr

1. Introduction

Because turbulent flow through tube bundles can be found in many important industrial applications, such as PWR reactor, steam generator, CANDU calandria and lower plenum of the VHTR, extensive studies have been made both experimentally and numerically. Although recently licensing applications supported by commercial CFD software are increasing, there is no commercial CFD software which obtains a licensing from the regulatory body until now. Therefore, it is necessary to perform the systematic assessment for the prediction performance of the commercial CFD software. The main objective of the present study is to numerically simulate turbulent flow through both staggered [1] and in-line tube bundle [2] using the two popular commercial CFD software, ANSYS CFX [3] and FLUENT [4] and to compare the simulation results with the experimental data for the assessment of these software’s prediction performance.

2. Numerical Method and Results

2.1 Staggered tube bundle

The experimental data of Paul et al. [1] for turbulent flow through the staggered tube bundles are used for the comparison. As shown in Fig. 1, the staggered tube bundles consist of 6 rows of tubes of outer diameter of 25.4 mm. The longitudinal distance between two tubes is 53.34 mm and the transverse distance is 96.52 mm. The longitudinal distance between inlet and first row of tube is 1.116m.

The flow is assumed to be steady, incompressible and turbulent. Uniform velocity with the magnitude of 0.34 m/s which correspond to Reynolds number 9,300 is imposed at inlet boundary. Turbulence intensity at inlet is set to be 4%. At the outlet boundary, static pressure is specified. No-slip condition is applied on the solid wall.

Table I: Summary of the numerical modeling

<table>
<thead>
<tr>
<th>Items</th>
<th>ANSYS CFX</th>
<th>FLUENT</th>
</tr>
</thead>
<tbody>
<tr>
<td>Turbulence model</td>
<td>RNG k-ε &amp; k-ω</td>
<td></td>
</tr>
<tr>
<td>Wall treatment</td>
<td>Scalable wall function</td>
<td>Enhanced wall treatment</td>
</tr>
<tr>
<td>Convergence criteria</td>
<td>≤ 10⁻⁶</td>
<td></td>
</tr>
<tr>
<td>Convection term</td>
<td>2nd order</td>
<td></td>
</tr>
</tbody>
</table>

Because of the symmetric geometry (y-direction), the computational domain considers only half of the experimental domain with symmetry boundary condition. Two different turbulence models, that is, RNG (ReNormalization Group) k-ε model and standard k-ω model are used. Table I shows the summary of the numerical modeling. A total number of cells with multi-block structured hexahedral shape are 643,200.

Fig. 2 and 3 show the comparison of the streamwise and transverse mean velocity profile at the selected axial locations. For the streamwise velocity a combination of FLUENT and RNG k-ε model gives the superior prediction performance to a combination of ANSYS CFX and RNG k-ε model except at x/d=7.15. In case of k-ω model, ANSYS CFX predicts well in the developing flow region, whereas FLUENT does well in the developed flow region.

![Fig. 1. Schematic diagram of staggered tube bundle test rig](image1.png)

![Fig. 2. Comparison of streamwise mean velocity profile at the selected axial locations](image2.png)
In this study, numerical analysis of turbulent flow through both staggered and in-line tube bundle using the two popular commercial CFD software, ANSYS CFX and FLUENT, was conducted and the simulation results were compared to experimental ones to assess the prediction performance of those software. The major conclusion could be summarized as follows:

1) Simulation results showed the large difference with the measurement especially for the in-line tube bundle.

2) FLUENT showed the overall superior prediction performance to ANSYS CFX for the converged solution.

### 3. Conclusions

The comparisons of the experimental and calculated pressure drops ($\Delta p = P_1 - P_3$) are summarized in Table II. Difference between the measurement and the prediction is above about 21.7%. These differences increase significantly in comparison with previous study [5] with 1st order upwind scheme for convection term and less dense grid. A combination of FLUENT and k-ω model shows the best prediction performance.

#### Table II: Comparison of the magnitude of pressure drop

<table>
<thead>
<tr>
<th>Item</th>
<th>Exp.[2]</th>
<th>ANSYS-CFX [k-ω]</th>
<th>FLUENT [k-ω]</th>
<th>RNG k-ω</th>
<th>RNG k-ε</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\Delta p$ [Pa]</td>
<td>28.2</td>
<td>17.6</td>
<td>16.2</td>
<td>22.1</td>
<td>19.8</td>
</tr>
<tr>
<td>Error [%]</td>
<td>-</td>
<td>37.6</td>
<td>42.7</td>
<td>21.7</td>
<td>29.9</td>
</tr>
</tbody>
</table>

Note: Error [%] = (Exp.-Comp.)/Exp. x 100

### Acknowledgement

This study was conducted under the financial support of the National Research Foundation of Korea [project title: Development of Safety Evaluation Capability on New Design Features]. The authors also gratefully thank Mr. Kang Dong-Gu for valuable comments.

### REFERENCES


