OpenFOAM simulations of the Turbulent Flow in a Rod Bundle with Mixing Vanes

Blaž Mikuž
Reactor Engineering Division, Jozef Stefan Institute, Jamova cesta 39
SI-1000 Ljubljana, Slovenia
blaz.mikuz@ijs.si

Samo Košmrlj, Iztok Tiselj
Reactor Engineering Division, Jozef Stefan Institute, Jamova cesta 39
SI-1000 Ljubljana, Slovenia
samo.kosmrlj@gmail.com, iztok.tiselj@ijs.si

ABSTRACT

Fuel assembly of a PWR reactor core is cooled by pressurized water, which flows along the fuel rods and removes heat. The heat transfer from the hot surfaces of the fuel rods to the surrounding coolant stream is better if the mixing in the flow is enhanced. Effective mixing devices are mixing vanes. They are attached to a spacer grid which fixes the rods in a fuel assembly and deflect the flow to produce turbulent flow behind them. Several types of mixing vanes are currently being designed, which can mix the flow effectively and with low pressure losses. The performance of mixing vanes was previously examined through experimental work which requires a lot of efforts to get reliable data. The other approach is with Computational Fluid Dynamics (CFD), which can also provide additional information that cannot be measured in a real experiment.

In this paper a steady-state RANS simulation of the flow in the sub-channels of a $5 \times 5$ cold rod array with split-type mixing vanes has been performed and analysed using open-source OpenFOAM (Open Source Field Operation and Manipulation) software. In this way we simulated the experiment at the MATiS-H facility of the Korea Atomic Energy Research Institute (KAERI) in Daejeon (S. Korea) where the cross-sectional velocity fields along the fuel assembly were measured at the condition of $Re = 50250$ in the water loop being operated at $35^\circ C$ and 1.5 bar. Furthermore, the results from OpenFOAM steady-state simulation are compared with the results from ANSYS CFX transient simulation.

1 INTRODUCTION

Heat removal from the surface of fuel rods to the surrounding water can be enhanced if flow is more turbulent [1]. A simple way of enhancing the turbulence of the flow through fuel assembly is use of various types of mixing vanes (Fig.1). They are attached on spacer grid and deflect the flow, which produces swirls behind them. The position and shape of the mixing vanes must be optimized for good mixing at low pressure drop. In the past, many experiments have been done to find the optimum spacer grid. In the last decade the development of computer hardware and software enabled faster 3D simulations using different CFD codes. Now we have available commercial CFD codes, which are well tested and user-friendly and on the other side also non-commercial CFD codes, which are free and open-source. Such an open-source CFD
program is OpenFOAM, which allows us to develop new codes but on the other hand it is less trustworthy. Although CFD is becoming a very powerful tool, there are still no CFD codes available for sufficiently accurate estimation of the flow structure in the sub-channel geometry of spacer grid. Therefore better CFD codes are being developed by improving the turbulence model and the numerical schemes and their results must be validated with experimental data. For this paper a CFD simulation of turbulent flow through $5 \times 5$ fuel bundle with split-type spacer grid (Fig.1a) has been performed with non-commercial OpenFOAM software and compared with results from ANSYS CFX transient simulation done by S. Košmrlj.

2 MATIS-H EXPERIMENT

MATiS-H is an acronym for Measurement and Analysis of Turbulent Mixing in Subchannels - Horizontal. It is a test facility located at KAERI Institute which schematic is illustrated in Fig.2. The main parts of the MATiS-H facility are storage tank (e), circulation pump (f) and...
test section (a) with Laser Doppler Velocimetry (LDV) probe (l). The water in storage tank is accurately maintained at constant temperature by controlling the heater (i) and the cooler (h) while the flow rate in the loop is controlled by adjusting the rotational speed of the pump. In the test section a $5 \times 5$ rod bundle array (p) is installed in a horizontal position. The water flow enters the test section and is straightened by two flow straighteners. The first flow straightener (d) is placed before rod bundle array while the second one (d) is located in rod bundle array at sufficient distance upstream to allow the flow to fully develop. After that, water flows through spacer grid with mixing vanes where turbulent mixing is dramatically enhanced. The main purpose of the MATiS-H experiment was to obtain accurate measurements of the cross-flows in the subchannels at various downstream locations from the spacer grid [2]. In order to improve the measurement resolution with LDV probe, the rods have 2.67 times larger diameter than the real size PWR fuel rods. The other lengths are also consistently multiplied by the same factor and summarized in Table 1 together with the flow parameters.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Mean Value</th>
<th>Overall Uncertainty (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rod-to-Rod Pitch ($P$)</td>
<td>33.12 mm</td>
<td>/</td>
</tr>
<tr>
<td>Wall-to-Rod Pitch</td>
<td>18.76 mm</td>
<td>/</td>
</tr>
<tr>
<td>Hydraulic Diameter ($D_{H}$)</td>
<td>24.27 mm</td>
<td>/</td>
</tr>
<tr>
<td>Total length of test section</td>
<td>4970 mm</td>
<td>/</td>
</tr>
<tr>
<td>Mass Flow Rate</td>
<td>24.2 kg/s</td>
<td>0.29</td>
</tr>
<tr>
<td>Temperature</td>
<td>35°C</td>
<td>2.9</td>
</tr>
<tr>
<td>Pressure</td>
<td>1.569 bar</td>
<td>0.39</td>
</tr>
<tr>
<td>Bulk Velocity ($W_{bulk}$)</td>
<td>1.50 m/s</td>
<td>0.37</td>
</tr>
<tr>
<td>Reynolds Number</td>
<td>50250</td>
<td>2.01</td>
</tr>
</tbody>
</table>

### 3 CFD SIMULATION

#### 3.1 Mesh

The mesh was prepared with ANSYS ICEM CFD mesh generation software. Only the fluid domain was meshed since we are currently interested only in single-phase water flow without heat transfer. The same mesh was used also for transient simulations with ANSYS CFX program (case study done by S. Košmrlj [3]) and here both results are compared with measurements from MATiS-H facility. To reduce the computational demands, only the section from $10D_{H}$ upstream from the spacer grid to $10D_{H}$ downstream of the spacer grid shown in Fig 3a was modelled. Geometry around mixing vanes is better shown in Fig 3b. The blue colour on that figure correspond to regions where water flows. The whole mesh consists of cca 13.1 million elements, which are all hexahedral to speed up the convergence of simulation [4]. It was stored in fluent format (ANSYS Fluent) and converted to OpenFOAM format. Although the maximum non-orthogonality of the mesh was almost 88°, an average non-orthogonality was only 14.6° and a good convergence was achieved.

---

1 A mesh is orthogonal if, for each face within it, the face normal is parallel to the vector between the centres of the cells that the face connects, e.g. a mesh of hexahedral cells whose faces are aligned with a Cartesian coordinate system [5].
3.2 OpenFOAM simulation

A steady-state RANS simulation of the turbulent flow in a rod bundle with split-type mixing vanes was performed with OpenFOAM (Open Source Field Operation and Manipulation). The steady state solution was calculated with \textit{simpleFoam} solver, which is an implementation of \textit{SIMPLE} algorithm with no \( \partial/\partial t \) terms. It solves the mass continuity equation and Navier-Stokes equation for incompressible flow:

\[
\nabla \cdot \mathbf{U} = 0 \quad \quad \nabla \cdot (\mathbf{UU}) - \nabla \cdot ((\nu_k + \nu_{turb}) \nabla \mathbf{U}) = -\frac{\nabla p}{\rho} \tag{1}
\]

where \( \nu_k \) is kinematic viscosity, which was calculated at \( p_0 = 1.5 \text{ bar} \) and \( T_0 = 35^\circ \) with XSteam program [6]. It gives dynamic viscosity \( \eta \) and density of liquid water \( \rho \):

\[
\eta(p_0, T_0) = 7.19 \cdot 10^{-4} \text{ Pa s} \quad \quad \rho(p_0, T_0) = 994 \text{ kg/m}^3 \Rightarrow \nu_k = \frac{\eta}{\rho} = 7.24 \cdot 10^{-7} \text{ m}^2/\text{s}
\]

The \( \nu_{turb} \) is turbulent kinematic viscosity computed with \( k - \omega \) SST turbulent model described below. Implementation of Eq. (1) in \textit{simpleFoam} solver is performed by derivation of an equation for the pressure using the divergence of the momentum equation. Knowing \( p \), a momentum and flux correction are performed using continuity equation. Iterations over momentum and pressure equations can be summed up as follows:

- Boundary conditions setup.
- Momentum equation solved with old pressure field in order to compute the intermediate velocity field.
- Compute the mass fluxes at the cells faces.

\footnote{It is an acronym for Semi-implicit methods pressure-linked equations.}
• Solve the pressure correction equation and apply under-relaxation.
• Correct the mass fluxes at the cell faces.
• Correct the velocities on the basis of the new pressure field.
• Boundary conditions update.
• Repeat until convergence.

For discretization of gradient, divergence and Laplacian terms, a second order, central differencing scheme is used.

A RANS (Raynolds-Averaged Navier Stokes) simulation is performed using $k - \omega$ SST turbulent model as the most suitable for separating flows [7]. It is a two-equation eddy-viscosity model introducing equations for turbulent kinetic energy $k$ and specific dissipation rate $\omega$. It combines the best of two worlds: $k - \omega$ formulation is usable in the inner parts of the boundary layer while the SST formulation switches to a $k - \epsilon$ behaviour in the free stream and thereby avoids the common $k - \omega$ problem that the model is too sensitive to the inlet free-stream turbulence properties. However, it is believed that $k - \omega$ SST model produces a bit too large turbulence in regions with large normal strain (e.g. stagnation regions) and regions with strong acceleration.

3.3 Boundary conditions

Boundary conditions (BC) for four variables ($p$, $U$, $k$, $\omega$) have to be specified. For $p$ a zero-gradient BCs are chosen everywhere except on outlet, where fixed value of 1.569 bar is specified. Velocity field $U$ has fixed value of zero velocity on all surfaces. A uniform mass flow rate of $\phi_m = 24.2 \, kg/s$ (or corresponding volumetric flow rate of $\phi_V = \phi_m/\rho = 0.0243 \, m^3/s$) is specified on inlet, and a zero-gradient velocity BC is specified on outlet. Wall functions are used for $k$ and $\omega$ BC on surfaces and turbulent intensity of 5% is specified on inlet.

4 RESULTS

In the MATiS experiment a special interest was intended to measure time averaged values of velocity in three subchannels between the rods shown in Fig.4. Since we are interested in

![Figure 4: Measurement locations in MATIS-H experiment [3].](image)

the flow behind the mixing vanes, the origin of the coordinate system is in the geometric center of the $xy$ cross-plane shown in Fig.4 and at the tips of the mixing vanes in $z$-direction. For

---

3Shear stress transport.
convenience, the lengths in the \( xy \)-plane are scaled with respect to rod-to-rod pitch \( P \), while the lengths in \( z \)-direction (flow direction) are scaled with respect to the hydraulic diameter \( D_H \).

The convergence of steady state simulation with \texttt{simpleFoam} solver was typically in less than 500 steps which is relatively fast. It was run in parallel on 48 computer cores, which took around 1 hour. The solution of velocity magnitude field at distance 0.5 \( D_H \) behind mixing vanes is shown on Fig.5a. The symmetry of the velocity field on Fig.5a is due to the geometry of the mixing vanes, which cause swirls in the flow. The field of velocity magnitude along direction of flow is shown in Fig.5b, where \( xz \)-plane of velocity magnitude at \( y = 0.5 \ P \) is presented. A thin horizontal line is drawn in Fig.5b at distance 0.5 \( D_H \) behind mixing vanes indicating the position of \( xy \)-plane from Fig.5a.

In the MATiS-H experiment, time averaged values were measured along the lines at 0.5 \( D_h \), 1.0 \( D_h \), 4.0 \( D_h \) and 10.0 \( D_h \) downstream of the vane tips in three subchannels between the rod rows (Fig.4). The results of OpenFOAM steady-state simulations were collected along the same lines and compared with the results of transient simulation done with ANSYS CFX by S. Košmrlj ([3] and Fig.6). The blue curve on Fig.6 correspond to OpenFOAM results, while the red coloured curve correspond to CFX results. The best agreement is achieved in \( U_y \) velocity component, while the matching with results for \( U_x \) in \( U_z \) is worse. The positions of amplitudes are in a good agreement with the results of CFX simulation, while the magnitudes of amplitudes are typically overestimated. The experimental results are not shown here, but they are in a very good agreement with CFX results, whereas the matching with OpenFOAM results is worse. This may imply that there are some parts of the flow, where steady-state approximation is insufficient.
Figure 6: Comparison of velocity components between steady-state OpenFoam simulation (blue) and transient CFX simulation (red) at $y = 0.5 \, P$ and two different locations along $z$-direction: $z = 0.5 \, D_H$ (graphs (a),(c), (e)) and $z = 1.0 \, D_H$ (graphs (b),(d),(f)).
5 CONCLUSION

The results of steady state OpenFOAM simulation reasonably predicts the trends of velocity components while the magnitudes are overestimated. The results of transient simulation done with ANSYS CFX show much better level of agreement between computational model and actual experimental data. Therefore, the next step will be a transient simulation in OpenFOAM, which is expected to give better agreement with experiment than steady state simulation. Since the calculation will be performed on the same mesh, it will also give a better comparison of the two CFD codes: OpenFOAM versus ANSYS CFX.

ACKNOWLEDGMENTS

Blaž Mikuž was financially supported by the young researcher fellowship of the ministry of Ministry of Education, Science, Culture and Sport, Republic of Slovenia.

REFERENCES


