Large eddy simulation on the turbulent mixing phenomena in 3×3 bare tight lattice rod bundle using spectral element method

Haoran Ju a, Mingjun Wang a, *, Yingjie Wang a, Minfu Zhao b, Wenxi Tian a, Tiancai Liu b, G.H. Su a, Suizheng Qiu b, **

a State Key Laboratory of Multiphase Flow in Power Engineering, School of Nuclear Science and Technology, Xi'an Jiaotong University, Xi'an, 710049, Shaanxi, PR China
b China Institute of Atomic Energy, P. O. Box 275-59, Beijing, 102413, China

1. Introduction

With the rapid development of high performance computing and massive parallel techniques, thermal-hydraulic analysis for the fuel assembly in nuclear reactor core with CFD approaches is becoming an effective method to optimize the assembly design, making it possible reach an as low as reasonable available safety margin to maximize economic efficiency [1]. Currently, Reynolds averaged Navier Stokes (RANS) method is widely used in demonstration of T-H in reactor core and fuel assembly [2–4] as well as for flow mixing researches in pipes in nuclear systems [5,6]. Mixing devices such as mixing vane and wire wrap have been assembled in the core to reduce the local enthalpy peak caused by inhomogeneous power distribution [7]. Meanwhile, tight lattice geometry is broadly adopted in the advanced reactor due to the high conversion ratio and enhancement in turbulent mixing between adjacent subchannels. In bare tight lattice rod bundle, turbulent coherent structure causes strong transverse mixing flow between subchannels. The vortex street could be generated in the gap and be closely related to the mixing phenomena [8]. The gap vortex street [9] is a specific cross-flow pulsation phenomenon in the tight lattice bundle geometry which is tightly connected to turbulent coherent structures. Large-scale vortices could be transported across the gap, leading to momentum and energy exchange of the fluid on either side of the gap without net mass transfer. The detailed mechanism has been analyzed by Ref. [10]. To investigate the mixing principle and obtain calculation models, many research groups have...
conducted experimental studies on flows in tight lattice rod bundles with varieties of pitch-to-diameter ratio P/D [11], measured and analyzed turbulent flow patterns in rectangular duct containing 4 rod array and 6 rod cluster with P/D = 1.071 and 1.107 respectively [12]. Investigated the turbulent characteristics in wall channel in 37 triangular rod bundle assembled in P/D = 1.12 and 1.06 and studied phenomena in central channels [13]. Time mean velocities, axial-pressure distributions, wall shear stresses, Reynolds stresses and turbulent kinetic energy are focused in the experiments while time mean fluid temperatures, wall temperatures and turbulent heat fluxes were also measured in heated apparatus. For detailed coherent flow pulsation analysis [14,15], have also investigated Reynolds-averaged and phase-averaged variations in flow field. Large-scale, quasi-periodic structures across the gap regions have been demonstrated. More details related to the cross-flow pulsation are investigated [16] identified quasi-periodic flow pulsation near the gap regions and concluded the coherent structures highly interfere with each other in adjacent gap.

As for numerical studies on coherent patterns in the vicinity of rod bundle gaps, LES and URANS models have been employed to simulate flows in tight-lattice fuel bundles by Ref. [17]. Q-factor contour has been plotted to display the structure shape. Proper orthogonal decomposition is utilized to make insight into the coherent structures. Effect of the pitch to diameter ratio on turbulent flow pattern has also done with URANS method by Ref. [18]. [19] calculated turbulent flow in different geometries combining with periodic boundaries. Modified Qm factor is used to identify the shape of coherent structure in stream-wise direction.

Actually, one of the purpose to measure and simulate the single-phase turbulent mixing in tight lattice rod bundle is to work out an optimized method to predict the mixing phenomena and apply the model in subchannel code [20]. Currently, the most accepted model is the turbulent mixing coefficient, which defines as follows [21]:

$$
\beta = \frac{U_{ij,eff}}{C_{ij,bulk} S_{ij}}
$$

where $\beta$ expresses turbulent mixing coefficient ($-$) $U_{ij,eff}$, effective lateral fluctuating flow rate per unit length between subchannel $i$ and $j$ (kg/m·s), $C_{ij,bulk}$, average axial bulk mass velocity (kg/m²·s) and $S_{ij}$, gap width (m).

[22] gave a theoretical estimation to show the correlation for Reynolds number dependence in form of $\beta = C Re^m$, where $m = -0.1$ when the Reynolds numbers reaches $2 \times 10^4$. Subchannel code such as COBRA IV has already taken Re-dependent turbulent mixing effect into account [23], demonstrated the same conclusion with high accuracy experimental result for Re larger than $3 \times 10^4$. A summary of the parameters for the turbulent mixing coefficient through adjacent subchannel with undisturbed flow conditions was laid out in Table 1 [21]. The results express that the turbulent mixing coefficient is not only related to Reynolds number, but also geometry parameters like pitch to diameter ratio P/D and hydraulic diameter of a subchannel to gap as well.

However, turbulent mixing intensities have not fully investigated in different places in rod bundle to work out the influences by geometry. In this paper, high-order CFD approach is employed to study turbulent mixing phenomena in $3 \times 3$ bare rod array with large eddy simulation model. The flow pulsation through gap in different positions are detected. Lateral velocities at gap center between corner channel and wall channel, wall channel and wall channel, wall channel and center channel as well as center channel and center channel are collected and compared with each other.

<table>
<thead>
<tr>
<th>Channel type/medium</th>
<th>P/D</th>
<th>$D_h$ (mm)</th>
<th>$\beta$</th>
</tr>
</thead>
<tbody>
<tr>
<td>S-T/water</td>
<td>1.036</td>
<td>5.105</td>
<td>0.063 Re $^{-0.1}$</td>
</tr>
<tr>
<td>S-S/water</td>
<td>1.149</td>
<td>7.290</td>
<td>0.021 Re $^{-0.1}$</td>
</tr>
<tr>
<td>S-S/air</td>
<td>1.334</td>
<td>11.56</td>
<td>0.027 Re $^{-0.1}$</td>
</tr>
<tr>
<td>S-S/air</td>
<td>1.100</td>
<td>27.10</td>
<td>0.02968 Re $^{-0.1}$</td>
</tr>
<tr>
<td>S-S/air</td>
<td>1.375</td>
<td>57.30</td>
<td>0.01683 Re $^{-0.1}$</td>
</tr>
<tr>
<td>S-S/air</td>
<td>1.833</td>
<td>125.0</td>
<td>0.009225 Re $^{-0.1}$</td>
</tr>
<tr>
<td>S-S/air</td>
<td>-</td>
<td>-</td>
<td>0.004 $\left(\frac{D_h}{s}\right)$ Re $^{-0.1}$</td>
</tr>
<tr>
<td>T-T/air</td>
<td>1.011</td>
<td>10.57</td>
<td>2.8365 $\times 10^{-13}$ Re $^{-4.41}$</td>
</tr>
<tr>
<td>T-T/water</td>
<td>1.028</td>
<td>11.17</td>
<td>0.001571 Re $^{-0.23}$</td>
</tr>
<tr>
<td>T-T/water</td>
<td>1.063</td>
<td>12.42</td>
<td>0.002871 Re $^{-0.12}$</td>
</tr>
<tr>
<td>T-T/water</td>
<td>1.127</td>
<td>14.69</td>
<td>0.002277 Re $^{-0.12}$</td>
</tr>
<tr>
<td>T-T/water</td>
<td>1.228</td>
<td>18.29</td>
<td>0.005999 Re $^{-0.01}$</td>
</tr>
<tr>
<td>T-T/water</td>
<td>-</td>
<td>-</td>
<td>0.005 $\left(\frac{D_h}{s}\right)\left(\frac{P}{d}\right)$ Re $^{-0.1}$</td>
</tr>
<tr>
<td>T-T/water</td>
<td>0.400</td>
<td>29.4</td>
<td>0.007479 Re $^{-0.1}$</td>
</tr>
<tr>
<td>T-T/water</td>
<td>0.100</td>
<td>12.73</td>
<td>0.007 Re $^{-0.065}$</td>
</tr>
</tbody>
</table>

The revised method for turbulent mixing correlation is also suggested for application in subchannel code.

2. Numerical methods and case settings

2.1. Governing equations

In this study, the high-order open-source CFD code Nek5000 [24] is utilized to perform large eddy simulations based on spectral element method (SEM) [25]. Currently Nek5000 is widely used in thermal-hydraulic analysis in fuel assembly in PWR [1,26] and SFR [27,28]. As a high-order weighted residual approach integrating finite element method for geometric flexibility and the tensor product efficiencies of spectral method, SEM may have the potential capability to overcome the limitations of standard CFD solvers in long term of unsteady calculation. Owing to recent rapid development of HPC and parallel computing technique, turbulent models with high precision such as LES has become an enhancing affordable calculating approach in CFD analysis, which could also accompany well with high-order numerical method.

In Nek5000, turbulent flow is governed by 3D non-dimensional incompressible Navier-Stokes equation

$$
\frac{\partial u}{\partial t} + u \cdot \nabla u = -\nabla p + \frac{1}{Re} \nabla^2 u + f, \quad \Omega \subset \mathbb{R}^3
$$

with the divergence-free velocity constraint

$$
\nabla \cdot u = 0
$$

where $\mathbf{u} = (u, v, w)$ stands for the velocity, $p$ is the pressure, $\mathbf{f} = (f_1, f_2, f_3)$ represents forcing functions The variables concerned in equations (2) and (3) are nondimensionalized with the characteristic parameter in the flow field [25]: $x = \frac{x}{L}, \quad u = \frac{u}{U_0}, \quad v = \frac{v}{U_0}, \quad f = \frac{f_D}{U_0^2}$.

Considering the dominance of the convective effects in this case, the pressure scale goes to $p \approx \frac{\rho D_h}{\nu}$. The subscript $f$ and $b$ indicate parameters in flow field and bulk flow, respectively. With the nondimensionalized method above, the dynamic viscosity $\mu$ finally replaced by the non-dimensional Reynolds number $Re = \frac{\rho D_h}{\mu}$.

Algebraic type of solution (PN-PN-2) and splitting scheme (PN-Neq-PN) are implemented in the code to solve the flow field. In this paper,
the splitting scheme (Orszag et al., 1986) is employed for the calculation. Detailed calculating methodology in splitting scheme will be introduced in this part. For simplicity, \( L(\mathbf{u}) \) and \( N(\mathbf{u}) \) denotes the linear and the non-linear terms in momentum equation and rewrite the equation as

\[
\frac{\partial \mathbf{u}}{\partial t} = \nabla p + N(\mathbf{u}) + L(\mathbf{u}) + \mathbf{f}
\]  

(4)

Totally, semi-implicit method is chosen to solve the unsteady advection-diffusion problem. \( k \)-th order backward-difference (BDF\( k \)) scheme with \( k \leq 3 \) is utilized for high-order time advancement.

\[
\sum_{j=0}^{k} b_j \Delta t^j \mathbf{u}^{n+j} = -\nabla p^{n+1} + N(\mathbf{u}^{n+1}) + L(\mathbf{u}^{n+1}) + \mathbf{f}^{n+1}
\]  

(5)

For nonlinear term \( N(\mathbf{u}) \), by explicit extrapolation we have

\[
N(\mathbf{u}^{n+1}) = \sum_{j=1}^{k} a_j N(\mathbf{u}^{n+1-j})
\]  

(6)

And different forms of linear term would be applied, the derivation procedure goes to

\[
L(\mathbf{u}) = \nabla \cdot [\mathbf{u} \mathbf{u}^T - \frac{2}{3} \nabla \cdot \mathbf{u}]
\]

\[
= \mu \left[ \Delta \mathbf{u} + \frac{1}{3} \nabla (\nabla \cdot \mathbf{u}) \right]
\]

\[
= \mu \left[ 4 \nabla (\nabla \cdot \mathbf{u}) - \nabla \times (\nabla \times \mathbf{u}) \right]
\]  

(7)

Calculating the force term explicitly \( \mathbf{f}^{n+1} \) and substituting the corresponding terms of Eq. (4) with Eq. (5) and Eq. (6), we have

\[
\frac{b_0}{\Delta t} \mathbf{u}^{n+1} = -\nabla p^{n+1} + L(\mathbf{u}^{n+1})
\]

\[
-\sum_{j=1}^{k} b_j \mathbf{u}^{n+1-j} + \sum_{j=1}^{k} a_j N(\mathbf{u}^{n+1-j}) + \mathbf{f}^{n+1}
\]  

\[
F(\mathbf{u}^n)
\]  

(8)

With the schemes above, the momentum equation is divided into 3 equations

\[
\frac{b_0}{\Delta t} [\mathbf{u}^{n+1}]^* = F(\mathbf{u}^n)
\]  

(9)

\[
\frac{b_0}{\Delta t} \left( \left[ \mathbf{u}^{n+1} \right]^* - \left[ \mathbf{u}^{n+1} \right] \right) = -\nabla p^{n+1}
\]  

(10)

\[
\frac{b_0}{\Delta t} \left( \left[ \mathbf{u}^{n+1} \right]^* - \left[ \mathbf{u}^{n+1} \right] \right) = L(\mathbf{u}^{n+1})
\]  

(11)

In the meantime, 3 steps have been designed for the splitting scheme implement:

- Nonlinear step. Eq. (9) is calculated depending on the initial condition at the beginning step or velocity at the previous time step.
- Pressure step. Eq. (10) is resolved in this step. To calculate the pressure gradient term on the right hand side of Eq. (10), we start from Eq. (5) with the derivation on the last line in Eq. (7)

Applying the divergence on Eq. (12)

\[
\nabla \cdot \nabla p^{n+1} = -\frac{b_0}{\Delta t} \nabla \cdot \mathbf{u}^{n+1} - \nabla \cdot \left( \frac{4}{3} \nabla (\nabla \cdot \mathbf{u}) + \mathbf{F}(\mathbf{u}^n) \right)
\]  

(13)

Realizing that \( \nabla \cdot \mathbf{u} = 0 \) is satisfied in any situations in incompressible flow, we could finally obtain the pressure Poisson equation and calculate the pressure

\[
\nabla \cdot \nabla p^{n+1} = \nabla \cdot \mathbf{F}(\mathbf{u}^n)
\]  

(14)

then, with the resolved pressure term \( p^{n+1}, [\mathbf{u}^{n+1}]^* \) could be obtained by Eq. (10).

- Viscous step. This is the final step of splitting scheme. Eq. (11) is solved in this step. With the resolved \( [\mathbf{u}^{n+1}]^* \) and the derivation on the 2nd line in Eq. (7)

\[
\frac{b_0}{\Delta t} \left[ \left[ \mathbf{u}^{n+1} \right]^* - \left[ \mathbf{u}^{n+1} \right] \right] = \mu \left[ \Delta \mathbf{u}^{n+1} + \frac{1}{3} \nabla (\nabla \cdot \mathbf{u}^{n+1}) \right]
\]  

(15)

Using \( \nabla \cdot \mathbf{u} = 0 \) again, we could reach the velocity Helmholtz equation from Eq. (11)

\[
-\mu \Delta \mathbf{u}^{n+1} + \frac{b_0}{\Delta t} \mathbf{u}^{n+1} = \frac{b_0}{\Delta t} \left[ [\mathbf{u}^{n+1}]^* \right]
\]  

(16)

and get the latest \( \mathbf{u}^{n+1} \) with velocity boundary condition.

High-efficiency algorithm has been adopted in the NEEK5000. For temporal integration, characteristic scheme suggested by Ref. [29] is used to enlarge the Courant number and keep a transient stability during the time-marching approach, which could significantly reduce calculation steps for a certain transient analysis. As for spatial discretization, spectral element method (SEM) is used to solve the Helmholtz equation for velocity and the Poisson equation for pressure. Derived from finite element method, computational domain would be separated into pieces with collocation points set in each element.

Large eddy simulation model with high-pass filters technique
eddy-viscosity models to improve the quality of the prediction of turbulent flow parameters for modeling the SGS prediction. Calculation results given by LES with adequate fine mesh could be quite similar with a DNS solutions [33]. proposed that Δx = 15–30, Ny = 10–30 as well as Δz = 50–130 is a recommended grid resolution for wall-resolved LES model, where x, y, z represents spanwise, wall normal and streamwise direction. Fig. 1 exhibits the profile and the detail mesh in cross-sectional view of P/D = 1.06 geometry. For the case Re = 20500, the height of first off-wall mesh layer was set to reached a maximum y* value of 0.89. The number of mesh point on wall-normal direction is 29 since 8 collocation points lay on one direction in each spectral element. The area-averaged wall friction velocity, ur = \sqrt{\tau_w/\rho}, is 0.118 m/s while the Reynolds number calculated by friction velocity, Rez = uDz/ν = 1058. The spanwise resolution Δ(Rh) + = uDz/ν = 132 if the axial length of elements set to 1Dh, namely 0.818 mm. In the estimation process of (Re (Rh)) + and Δz, the mesh point distance in each spectral element is regarded as equidistant for convenience. Considering the Runge’s phenomenon, the collocation points based on Gauss-Legendre-Lobatto points could definitely get a much higher precision than the result given by equidistant points, which could also receive a proper grid resolution.

Time advancement is another ‘one-way’ mesh in large eddy simulation case. The 3rd order backward difference scheme is called in the transient simulation. With the superiority of OIFS method, the target Courant number is set to 3 to obtain the instantaneous flow field rapidly. The 10 flow through time (FFT) is paid to reach the statistical fully-developed turbulent flow condition from the initial flow field. Then another 10 FFT is calculated for data collection. For the convergence criteria, relative error at 10⁻¹ has been asked for velocities to highlight the advantage of spectral element method. Considering the sensibility of pressure in calculating progress, relative error 10⁻₆ has been set for its iteration.

### 3. Calculating results and discussions

#### 3.1. Profile of flow distribution

The instantaneous normalized stream velocity profile has been displayed in Fig. 2. The gap vortex street could be observed clearly along with the axial direction. Eddies in the subchannels transferred from one subchannel to another through the rod gap. More flow details could be obtained in Fig. 3. The cross-sectional flow profile illustrated velocity distribution in different subchannels. Center channels, which surrounded by 4 rod quarters, got the velocity peak and occupied much kinetic energy. Wall channels, formed by 2 rod quarters and square duct wall, sustained a medium-level velocity magnitude profile. Corner channels, restricted by 1 rod quarter and the wall corner, keepeled the eddies.

---

**Table 2**

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Case 1</th>
<th>Case 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rod diameter (mm)</td>
<td>25.4</td>
<td>24.0</td>
</tr>
<tr>
<td>Gap (mm)</td>
<td>0.75</td>
<td>1.45</td>
</tr>
<tr>
<td>P/D</td>
<td>1.03</td>
<td>1.06</td>
</tr>
<tr>
<td>Hydraulic diameter (mm)</td>
<td>6.63</td>
<td>8.02</td>
</tr>
<tr>
<td>Axial length (mm)</td>
<td>265.2</td>
<td>320.8</td>
</tr>
</tbody>
</table>

**Table 3**

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Re = 5000</th>
<th>Re = 10000</th>
<th>Re = 20500</th>
</tr>
</thead>
<tbody>
<tr>
<td>P/D</td>
<td>1.03</td>
<td>1.06</td>
<td>1.03</td>
</tr>
<tr>
<td>Kinematic viscosity (m²/s)</td>
<td>8.94e-7</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Density (kg/m³)</td>
<td>996.89</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Characteristic velocity (m/s)</td>
<td>0.674</td>
<td>0.557</td>
<td>1.35</td>
</tr>
<tr>
<td>Characteristic time (s)</td>
<td>9.84e-3</td>
<td>1.44e-2</td>
<td>4.92e-3</td>
</tr>
</tbody>
</table>
Fig. 1. Mesh of calculation domain.

a) Cross-sectional profile of mesh

b) Mesh details (mesh in the red box in a)

Fig. 2. Flow distribution of the instantaneous streamwise velocity.

Fig. 3. Flow field details on cross sections in rod bundle.
with lowest velocity value in the flow domain. The last 2 subgraphs down in Fig. 3 displayed gap vortex stress particulars at different rod gaps. Qualitatively, gaps related to center channel contains more coherent structures than those near the duct wall. Strong flow pulsation caused by frequent coherent structure made it tough to observe the fluctuating cycle directly in center gaps while in near-wall gaps it is much more clear: It could be counted that the number of vortex structures along the Z-axis varied at different gaps. 7 pulsation cycles have been contained in the 40$D_h$ long calculation domain through the wall-wall gap and only 6 cycles accrued at wall-corner gap. With the differences on turbulent mixing at various gaps, more qualitative analysis should be conducted on the transverse velocity at each gap in rod bundle.

3.2. Turbulent mixing phenomena analysis

Turbulent mixing through gaps is a specific phenomena in rod bundle geometries. Transverse fluctuating velocity at gap centers are monitored to specify turbulent mixing with different geometric conditions. Simulations and measurements have been performed on the related researches just at the gap center connecting center channel and center channel. However, Comparison among gaps between different kinds of subchannels has not been highly focused, which may also be crucial in safety analysis. With such a perspective, 4 types of gaps in $3 \times 3$ rod bundle have been selected to monitor turbulent mixing in this research, as shown in Fig. 4, namely.

![Fig. 4. Monitoring locations for transverse fluctuating velocity.](image)

![Fig. 5. Transverse fluctuating velocity results at different gap centers](image)

Fig. 4. Monitoring locations for transverse fluctuating velocity.

Fig. 5. Transverse fluctuating velocity results at different gap centers.
Fig. 5 demonstrates the mixing effect through different gaps in rod bundle with diverse Re number. It could be perceived that quasi-periodic flow, caused by turbulent pattern, occurred through each gap in rod bundle. Both of the fluctuating amplitude and period in each graph rise with the growing Re number. However, turbulent mixing characteristics separate slightly with different positions: Fluctuating frequency decreased sharply from center channel to corner channel. The instantaneous velocities were also flattened in the gap near duct wall.

More researches have been conducted for quantitative analysis. Quasi-periodic flow properties have been studied based on the Fast Fourier Transform (FFT) technique. The analysis results have been displayed in Fig. 6. The peak frequency $f_p$, which gained the peak power spectral density in each curve, figured out the frequency domain properties of turbulent mixing at rod gaps. The value of $f_p$ went down as the location of the gap approach to the wall and corner. The results calculated in the other cases with the have been listed in Table 4. Numbers in brackets are the relative $f_p$ values compared with the C–C channel value, which usually be attached importance and utilized in subchannel code.

Qualitatively, the peak frequencies at C–W gaps are quite close to that at C–C gaps compared with the W–Co and W–W ones. According to the theory proposed by Refs. [10], the quasi-periodic flow is governed by the vortex generated in subchannels on both sides of the gaps. Instantaneous velocity vectors, which could capture eddy distribution in rod bundle, have been plotted to illustrate secondary flow on cross-sectional area. Global and local distributions are printed in Fig. 7. It is worth to point out that the center channel keeps a wide range in eddy size with high velocity magnitude. Meanwhile, low-speed tiny eddies in similar length scales dominate the corner channel. Eddies could form a cascade and make effects on mixing through subchannels. Covering cascades formed by eddies with different length scale, center channels could send out fluctuating transverse flow to adjacent subchannel with a certain complicated flow details, as shown in Fig. 8 (a). In contrast, due to the near-wall narrow geometry, wall channels and corner channels could only offer cascades assembled by small eddies and provide simple fluctuating track as Fig. 8 (b).

The fluctuating period of eddy cascade is directly related to the
rotation frequency of eddies in the cascade. Obviously, eddies could receive a higher angular velocity and rotation frequency driven by large bulk velocity and rotate slowly in low-speed velocity field. Thus, turbulent mixing could gain a less period in high-speed channel than it in low-speed channel, as P1 and P2 noted in Fig. 8.

For the $3 \times 3$ rod bundle, considering the calculation results given by Fig. 3, bulk velocities in center channels reached a major level, wall channel minor level and corner channel least level. With the driven bulk flow field, the size of vortex in each subchannel decreased from center channel to corner channel and produced fewer fluctuations at the relative gaps and reached the fluctuating frequency in Table 4. Thus, rising up the bulk flow magnitude in a certain subchannel is one proper way to raise the fluctuating frequency at channel gaps.

Turbulent mixing coefficient $\beta$ is another essential parameter to judge the mixing capacity at channel gaps quantitatively. Bringing momentum and energy exchange of fluid between adjacent subchannels without net mass transfer, the root mean square value of transverse flow velocity at gap center is used and defined as effective mixing velocity. Total bulk flow velocity is employed for the dimensionless number formation. The distribution of $\beta$ in Re $= 20500$ P/D $= 1.06$ case is figured in Fig. 9. Similar with the peak

---

**Table 4**

<table>
<thead>
<tr>
<th>P/D = 1.06</th>
<th>P/D = 1.03</th>
</tr>
</thead>
<tbody>
<tr>
<td>Re = 5000</td>
<td>Re = 5000</td>
</tr>
<tr>
<td>Point1</td>
<td>Point1</td>
</tr>
<tr>
<td>11.482 ($-7.66%$)</td>
<td>8.5618 ($-17.94%$)</td>
</tr>
<tr>
<td>Point3</td>
<td>Point3</td>
</tr>
<tr>
<td>7.4264 ($-40.27%$)</td>
<td>6.5447 ($-37.28%$)</td>
</tr>
<tr>
<td>Re = 10000</td>
<td>Re = 10000</td>
</tr>
<tr>
<td>Point1</td>
<td>Point1</td>
</tr>
<tr>
<td>20.275 ($-33.34%$)</td>
<td>24.864 ($-23.05%$)</td>
</tr>
<tr>
<td>Point3</td>
<td>Point3</td>
</tr>
<tr>
<td>13.522 ($-55.54%$)</td>
<td>19.883 ($-38.47%$)</td>
</tr>
<tr>
<td>Re = 20500</td>
<td>Re = 20500</td>
</tr>
<tr>
<td>Point1</td>
<td>Point1</td>
</tr>
<tr>
<td>46.279 ($-18.52%$)</td>
<td>56.796</td>
</tr>
<tr>
<td>Point3</td>
<td>Point3</td>
</tr>
<tr>
<td>34.700 ($-38.90%$)</td>
<td>36.362 ($-35.98%$)</td>
</tr>
</tbody>
</table>

---

Fig. 7. Secondary flow on cross-sectional area with instantaneous velocity vector.
frequencies, turbulent mixing coefficients also differ among C–C channel, Co–W channel and W–W channel in rod bundle. 23.4% of the value could be absent comparing with the C–C channel value, which always act as the standard value in the previous experiment to show turbulent mixing capability. The rest of results in P/D = 1.06 geometry have been listed in Table 5. The distribution of relative value of $\beta$ keep in agreement in the 3 cases. However, the absolute value in $Re = 5000$ is quite far away from the other cases. One possible reason is that the Reynolds number is not high enough to keep a fully developed turbulent pattern and the phenomena diverged from the other cases due to a different flow pattern. The results in $Re = 10000$ and $Re = 20500$ cases agreed with each other due to the similar distributions in both absolute and relative values.

According to the results above, a considerable reduction of $\beta$ is formed at Co–W channel, W–W channel, C–W channel gap compared with C–C channel gap value. Thus, the local effects should be treated in a proper way. A reasonable approach is to perform corrections directly on the $\beta$ correlations when building the turbulent mixing model in subchannel code, with the given result or with the methodology announced in this paper, for a high-precision subchannel prediction result.

4. Conclusions

In this work, CFD approach based on spectral element method is employed to predict turbulent mixing phenomena through gaps in 3 × 3 bare tight lattice rod bundle and investigate the flow pulsation through gap in different positions. The turbulent mixing characteristics in the rod bundle with different Re numbers and pitch to diameter ratios were studied in detail. Lateral velocities at gap center between corner channel and wall channel, wall channel and wall channel, wall channel and center channel as well as center channel and center channel are collected and compared with each other. The conclusions are listed as follows.

(1) Combining with the periodic boundary conditions, the calculation domain with relatively short length could be used for simulation in rod bundle. 400x is a proper axial length for the geometry in 3 × 3 rod bundle.

(2) $\Delta(R')^+$ and $\Delta z^+$ calculated with equidistant mesh point distance in each spectral element could be reasonable used for mesh resolution evaluation. With the results given by

---

Fig. 8. Turbulent mixing caused by cascade of eddies
a) Cascade with wide length scale in high velocity field
b) Cascade with narrow length scale in low velocity field.

Fig. 9. Turbulent mixing coefficients of transverse velocity at different gaps in Re = 20500 P/D = 1.06 case.
Table 5  
\( \beta \) and relative values in cases with \( P/D = 1.06 \) geometry.  

<table>
<thead>
<tr>
<th>Point1</th>
<th>Point2</th>
<th>Point1</th>
<th>Point2</th>
<th>Point1</th>
<th>Point2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Re = 5000</td>
<td></td>
<td>Re = 10000</td>
<td></td>
<td>Re = 20500</td>
<td></td>
</tr>
<tr>
<td>0.073 (−19.8%)</td>
<td>0.091</td>
<td>0.1894 (−16.7%)</td>
<td>0.2275</td>
<td>0.1977 (−11.5%)</td>
<td>0.2235</td>
</tr>
<tr>
<td>0.067 (−26.4%)</td>
<td>0.068 (−25.3%)</td>
<td>0.1720 (−24.4%)</td>
<td>0.1819 (−20.1%)</td>
<td>0.1712 (−23.4%)</td>
<td>0.1875 (−16.1%)</td>
</tr>
</tbody>
</table>

Ref. [33], for wall-resolved LES model, \( \Delta x^* = \Delta (R_0)^* \) is 15–30, \( N_p \approx 10–30 \) as well as \( \Delta z^* \approx 50–130 \) is a recommended grid resolution while the first layer height should less than 1 for flow detail capture.

(3) Turbulent mixing velocity has been monitored at 4 points at different gap center. Analysis in both frequency domain and time domain shows a distribution in rod bundle for peak frequency and turbulent mixing coefficient. The peak frequency values at \( W = Co \) channel are about 40%—50% lower comparing with the \( C = C \) channel value and the turbulent mixing coefficient \( \beta \) decreases around 25%. Thus, the local effects should be treated in subchannel analysis tools. Corrections for \( \beta \) should be performed in subchannel code at gaps related to wall channel and corner channel for a reasonable prediction result.

(4) The fluctuating frequency at gap is relative to eddy cascade in the either side of the subchannels. Subchannels containing eddies with wide length scales provide more complex fluctuating details while eddy cascades driven by large bulk flow offer higher fluctuating frequency at subchannel gap. As a result, eddy cascade should be considered carefully in detailed analysis for fluctuating in rod bundle.

Declaration of competing interest

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

Acknowledgements

This research has been supported by the National Key R&D Program of China (Grant No. 2017YFB0202300) and the National Natural Science Foundation of China (Grant No. 11705139).

References