Academic Journal of Science and Technology
ISSN: 2771-3032 | Vol. 15, No. 2, 2025

Numerical Simulation of Heat Transfer in A Vibrating Body Strengthened Tube

Zhi Chen, Yunqiao Luo

School of Central South University of Forestry and Technology, Changsha 410004, China

Abstract: To improve the heat transfer efficiency of the heat exchanger, we studied adding a moving vibrating body inside the pipeline to enhance the heat transfer efficiency. Based on the immersed boundary method and the application of OpenFOAM, we focused on enhancing heat transfer inside pipes with vibrating bodies, analyzed the forced convection heat transfer mechanism under fluid structure coupling, and further explored the influence of various parameter changes on the enhanced heat transfer performance inside pipes. Results show that the presence of vibrating body does indeed improve the heat transfer efficiency inside the tube in a degree, and different parameters can change this value. The motion of the vibrating body will change the motion path and flow velocity of the fluid, thereby increasing the average Nusselt number on the wall surface; The temperature difference between the wall of the pipeline and the fluid can greatly affect this heat transfer efficiency. Although the change in the average Nusselt number of the wall caused by the change in the shape of the vibrating body is not significant, it may be due to imperfect modeling in this simulation.

Keywords: Enhanced heat transfer; OpenFOAM; vibrating body; Nusselt number; immersed boundary method.

1. Introduction

High heat transfer efficiency is a fundamental requirement for electrical equipment, cooling systems, heat exchange systems, and various industrial applications. Researchers employ various methods to enhance heat transfer effectiveness. Previous studies[1-3]introduced surface roughness and vortex generators to improve heat transfer in pipe flow systems. Under given friction conditions, rib-roughened surfaces demonstrate superior convective heat transfer enhancement compared to sand-grain roughened surfaces in pipe flow. Zhu et al.[4] investigated the effects of airfoil and vane-type vortex generators on heat transfer enhancement in turbulent channel flow. They reported that combining longitudinal vortex generators with roughness elements could enhance heat transfer by 450%. We aim to strengthen heat transfer by introducing a moving object within the heat pipe to perturb fluid flow, establishing a fluid-solid coupled heat transfer numerical simulation model based on the OpenFOAM platform and IBM (Immersed Boundary Method) technology.

Numerical simulations have been applied to study the kinematics of flexible or rigid structures, their effects on fluid mixing and heat transfer, and parameter optimization under specified flow conditions. The Arbitrary Lagrangian-Eulerian (ALE) and Immersed Boundary Method (IBM) techniques are widely used to simulate fluid-flexible structure-thermal interactions. Among these, the IBM [11] is prioritized for such problems as it mitigates the difficulty of remeshing when objects deform or move within the fluid domain. In recent years, IBM has effectively simulated fixed and movable solid obstacles in computational domains with complex boundary conditions. This methodology encompasses proven approaches for simulating intricate moving geometries, including the Lagrange multiplier method [5], level-set method [6], fictitious domain-surface method [7], and volumetric penalty method [8,9]. IBM has been demonstrated as a practical and effective approach for simulating fluid-structure interactions in incompressible flows and has been successfully applied to a wide range of problems [10].

OpenFOAM is now widely used in academic research and engineering for fluid analysis. As a mature open-source software, it employs the Finite Volume Method (FVM) based on unstructured grids to discretize partial differential equations, capable of handling complex geometries. It enables simulations of various flows such as rotating machinery, multiphase flows, heat transfer, chemical reactions, and porous media.

2. Physical Model and Governing Equations

Since the physical model of calculating convective heat transfer for fluid-structure interaction is not complex, a simplified model of the heat exchanger is established using the modeling tools ICEM and SpaceClaim, which come with the ANSYS platform. The effect of the movement of the vibrating body in the tube on the heat exchange between the coolant and the hot wall was studied.

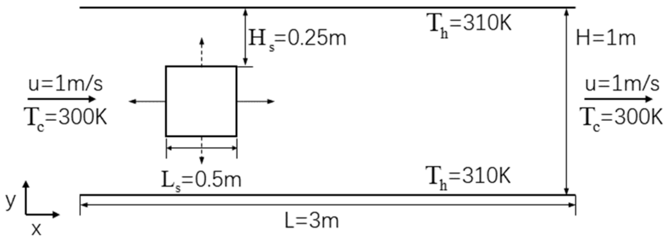

Figure 1. Physical model

As shown in Figure 1, the long lines on the upper and lower sides are the thermal walls on both sides of the pipe, the initial temperature is 310 K, the length is 3 m, and the inner diameter of the pipe is 1 m. The coolant has a temperature of 300 K and a flow rate of 1 m/s, with left in and right out, passing through a moving vibrating body. The vibrating body is a square with a side length of 0.5 m, and it is 0.5 m away from both sides of the pipe wall. The vibrating body can move in different directions with different degrees of periodic, and can also change the shape of the vibrating body, such as a diamond, a ring, or become larger and smaller. The fluid flow direction is in the x-direction, the vertical wall is in the y-direction, and the gravitational acceleration acts in the -y direction. Assuming that the size of the z-direction is large enough, the effect of the end effect on the flow is negligible, i.e., the flow and heat transfer of the fluid is two-dimensional. A no-slip flow boundary condition is applied to a solid wall. It is assumed that the flow field inside the tube occurs only in the laminar flow region, therefore, the N-S equation and the momentum equation are described as:

$$\nabla \cdot u = 0 \tag{1}$$

$$\frac{\partial u}{\partial t} + u \cdot \nabla u = -\nabla p + \mu \nabla^2 u + f \tag{2}$$

where u is the velocity vector, p is the pressure, f is the impressed force source term that gives the boundary a slip-free condition, and the volumetric average fluid velocity u at the inlet is the characteristic velocity.

The energy equation is as follows,

$$\frac{\partial T}{\partial t} + u \cdot \nabla T = \frac{\mu}{\mathrm{Pr}} \nabla^2 T + q \tag{3}$$

where q is the heat source of the immersed body. The dimensionless temperature is defined as $T=(T-T_0)/(T_w-T_0)$, and T_w and T_0 represent the wall temperature and the inlet fluid temperature, respectively. The wall is set to a constant temperature.

The fluid-structure interaction equation is as follows,

$$\frac{\partial^2 \mathrm{X}}{\partial t^2} = \frac{\partial}{\partial s}\left(\sigma \frac{\partial X}{\partial s^2}\right) - \frac{\partial^2}{\partial s^2}\left(\gamma \frac{\partial^2 X}{\partial s^2}\right) - F \tag{4}$$

X represents the position of the vibrating body, σ is the tensile force along the coordinate direction, γ is the bending stiffness, and F is the Lagrangian momentum exerted by the surrounding fluid on the vibrating body.

Since the heat transfer performance of the system can be expressed by the heat transfer rate at the surface of the heat source, the average Nusselt number is used as the criterion. The average Nusselt number is only related to the width of the shell of the hot wall and the thermal conductivity of the fluid, which is expressed by the following formula:

$$Nu_{hot} = -\int_0^{-1} \frac{\partial T}{\partial X}\Big|_{x=0} dY \tag{5}$$

where T is the average temperature gradient of the hot wall after the coolant flows through, and X is the width of the hot wall. The size of the average Nusselt number can be used as an evaluation criterion for the heat exchange rate, which is intuitive and contrasting.

3. Numerical Methods and Code Verification

On the block structure grid, the finite volume method is used to discretize the equation as the basic numerical solution. Two adjacent blocks are connected by unstructured factors to ensure better continuity. Time discretization is based on the first-order Euler implicit, but since the immersion boundary method can achieve a time-independent discretization scheme, it can be steadily extended to more accurate second-order OpenFOAM schemes. The velocity-pressure coupling is solved by the built-in solver pisoFoam, where the stable additional term for the mass flux in the pressure correction loop is set to 0. For simulations that currently involve low to medium Reynolds numbers and regular block structure meshes, preconditioning is not used. For all variables, the required precision is 10-7 at each time step. For the numerical experiments and the mesh presented in this paper, the solver achieves good results between stability and accuracy.

To verify the feasibility of the code, the results of this paper are compared with the new bonded enhanced heat transfer under laminar flow conditions, which is also selected to be numerically simulated under laminar flow conditions, and the analysis is shown in Table 1 below. As can be seen from Table 1, the margin of error is less than 4%, and the results obtained are roughly the same as those in the literature.

Table 1. Compassion of Nu between published results and this article

Re	Literature [1]	This paper	Error
400	15.8	16.07	0.0168
800	26.1	25.65	0.0175
1200	37.2	36.63	0.016
1600	39.5	38.00	0.0395

Under the initial conditions (T=310 K, Ra=109, β=3×10-3, μ=10-2), five different uniform grids 50×50, 100×50, 150×50, 150×100, and 200×100 were selected for grid independence verification, as shown in Figure 2 below, the difference between the values of each average Nusselt number (Nu) calculated from the selected mesh was less than 1%, so it was considered as grid independence. All subsequent calculations were performed on a uniform grid of 150×50.

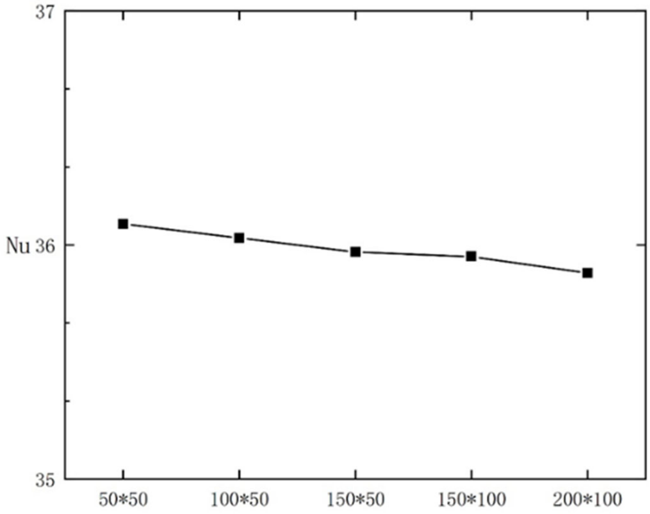

Figure 2. The influence of grid density on average Nusselt number

4. Results and Analysis

4.1. Effect of up-and-down vibration on heat transfer efficiency

In this subsection, the data under initial conditions are used to change only the amplitude of the upper and lower vibrations to explore the effect of the upper and lower vibrations on the heat transfer efficiency in the tube. Since the diameter of the tube is 1 m, under the realistic conditions, the amplitudes of six groups of vibrations are set sequentially, which are 0.05 m, 0.1 m, 0.15 m, 0.2 m and 0.25 m respectively. The resulting temperature contour and velocity vector plot are as follows:

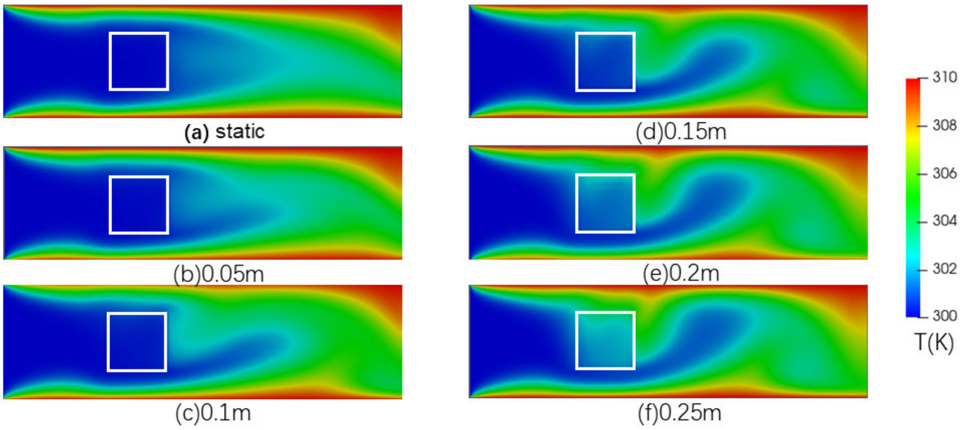

Figure 3. Temperature cloud map inside the tube under different vibration amplitudes of the vibration body under initial conditions

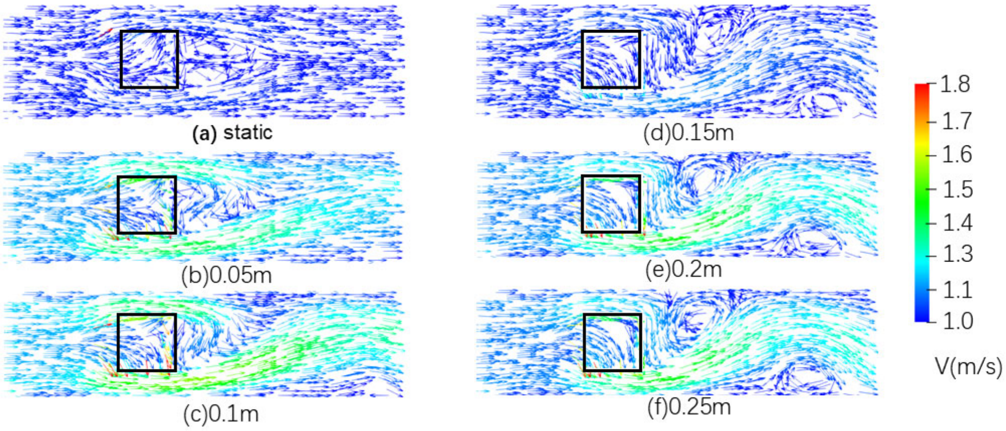

Figure 4. Velocity vector diagram inside the tube under different vibration amplitudes of the vibration body under initial conditions

As can be seen from Figures 3 and 4, the greater the amplitude of the upper and lower vibrations of the vibrating body, the higher the heat transfer efficiency. In the process of coolant flowing, the up and down vibration movement of the vibrating body will cause the path of coolant flow to change, resulting in the coolant constantly colliding with the wall, and the contact area will change, thus affecting the heat transfer efficiency. The coolant flow rate is stable at 1 m/s when the vibrating body is stationary, and the local coolant flow rate changes when the vibrating body vibrates up and down, which also makes a difference. The specific data is as follows:

Table 2. The influence of different up and down vibration amplitudes of a vibrating body on the Nusselt number under initial conditions

Vibration amplitude (m)	0	0.05	0.1	0.15	0.2	0.25
Nu	16.07	15.49	36.67	36.97	37.36	38

It can be seen from the table that the upper and lower vibrations of the vibrating body have a certain impact on the heat exchange efficiency, which increases the heat transfer efficiency by about double, but after the upper and lower vibrations reach a certain amplitude, the heat exchange efficiency will reach saturation.

4.2. Effect of wall temperature on heat transfer efficiency

This subsection uses the data under the initial conditions to explore the effect of the change of wall temperature on the heat transfer efficiency of the pipe under the condition of only changing the temperature of the inner wall of the pipe. Since the wall is always in a heat exchange process with the flowing coolant, the change in wall temperature greatly affects the efficiency of the heat exchange. Under realistic conditions, five different sets of wall temperatures (310 K, 320 K, 330 K, 340 K and 350 K) were set in order to explore the effect of wall temperature on heat transfer efficiency. Under the initial conditions, the temperature contour of the inner wall surface of the pipe, which only changes the temperature of the inner wall of the pipe, is shown below.

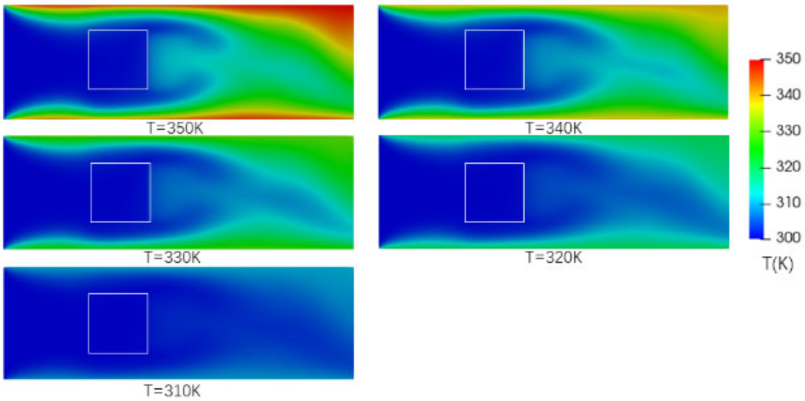

Figure 5. Comparison of temperature cloud maps inside pipes at different wall temperatures

As can be seen from Figure 5, the change of the initial temperature of the wall does not change the distribution of the temperature gradient, but does change the basic value of the temperature gradient. The higher the wall temperature, the higher the temperature of the coolant after the vibration of the vibrating body, the more intense the heat exchange.

To explore the effect of temperature change on the Nusselt number of the wall, as shown in Figure 6 below:

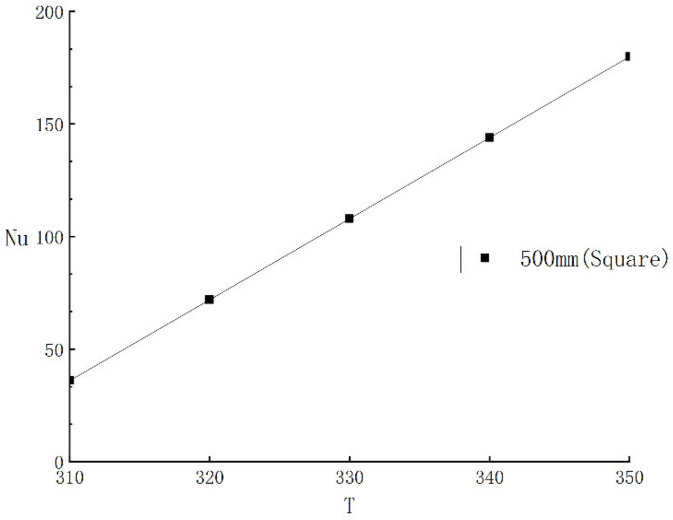

Figure 6. The influence of temperature changes on the average Nusselt number

In the case of changing only the wall temperature, the heat transfer efficiency in the tube increases as the wall temperature increases. The wall temperature has a great influence on the average Nusselt number, and the heat exchange between the wall and the boundary layer fluid occurs all the time during the flow process, and the initial base temperature of the wall determines the heat exchange efficiency. The average Nusselt number at 350 K is 143.9 higher than the average Nusselt number at 310 K, which means that the heat exchange efficiency is 400% higher. The specific data is as follows:

Table 3. The influence of different wall temperature on Nu

T(K)	310	320	330	340	350
Nu	35.97	71.95	108.18	143.9	179.87

4.3. Effect of vibrating body shape on heat transfer efficiency

This subsection uses the data under initial conditions to explore the influence of the change of the shape of the vibrating body on the heat transfer efficiency in the pipe under the condition of only changing the shape of the vibrating body in the pipe. Due to the different shapes of the vibrating bodies, the coolant will flow differently during the vibration process, and it is necessary to explore the influence of the shape of the vibrating body on the heat transfer efficiency. Under the condition of conforming to reality, four groups of different vibrating body shapes were set up in turn: square vibrating body (side length of 500 mm), diamond-shaped vibrating body, ring vibrating body and square vibrating body with side length of 700 mm. The specific data obtained are shown in Table 4 below:

Table 4. The influence of different shapes of vibrating bodies on Nu under different wall temperatures

T	500mm Square	Cirque	Lozenge	700mm Square
310K	35.97	35.99	35.9	36.31
320K	71.95	71.98	71.94	72.44
330K	108.18	107.97	107.91	108.66
340K	143.9	143.96	143.89	144.89
350K	179.87	179.95	179.86	181.11

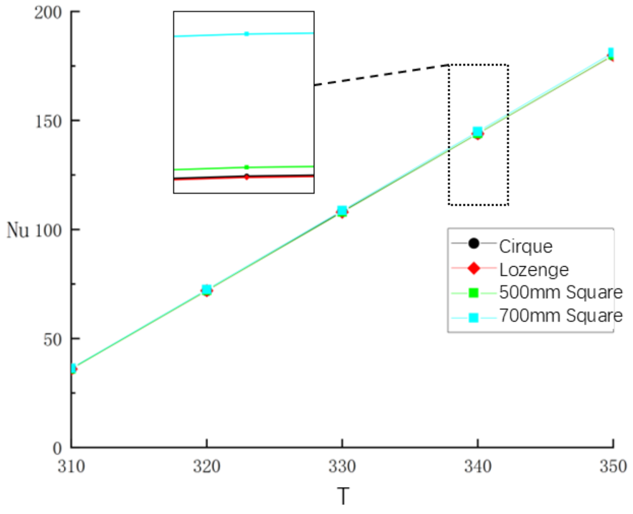

Figure 7. The influence of different shapes of vibrating bodies on the heat transfer efficiency inside the tube

The change in temperature has little effect on the heat exchange efficiency and almost coincides in Figure 7. It can still be seen that the Nusselt number is the largest under the 700 mm square vibrating body, followed by the 500 mm square and the ring, and the diamond is the smallest. Therefore, it can be inferred that, to a certain extent, the larger the volume or area of the vibrating body, the higher the heat transfer efficiency. It may be due to the small experimental simulation model, which leads to a small difference in the Nusselt number of the lower wall of various vibrating bodies, and this simulation data does not indicate that the shape of the vibrating body has little influence on the heat exchange efficiency of the wall.

4.4. Effect of Rayleigh number on heat transfer efficiency

In this section, the initial example data is used to explore the effect of the Rayleigh number on the heat transfer efficiency in the tube under the condition of changing only the Rayleigh number. The Rayleigh number is a dimensionless number used in fluid mechanics to describe natural convection. It is positioned as the product of the Glachev number and the Plante number. It reflects the competition between the buoyancy effect and the viscous effect, which determines how heat is transferred in the fluid.

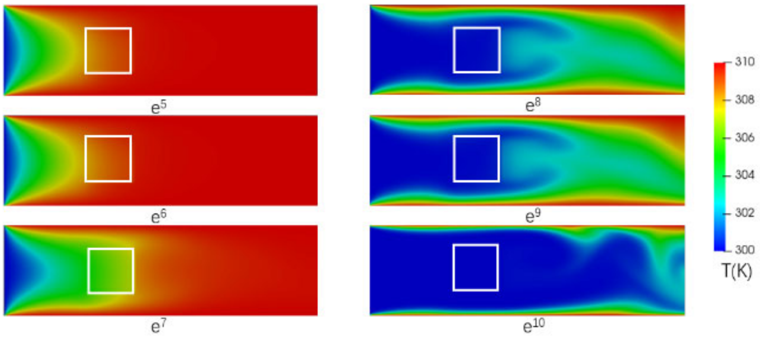

Figure 8. Temperature cloud map of pipe cross-section under different Rayleigh numbers

As can be seen from Figure 8, when the Rayleigh number reaches E7 and below, that is, the Rayleigh number is lower than the critical value, the main mode of heat transfer is heat conduction, and the efficiency of heat exchange in this state is very high, and all coolants can reach the wall temperature after the movement of the vibrating body. However, when the Rayleigh number exceeds E7, the main form of heat transfer becomes heat convection, and the heat transfer efficiency is not high, and decreases with the increase of Rayleigh number.

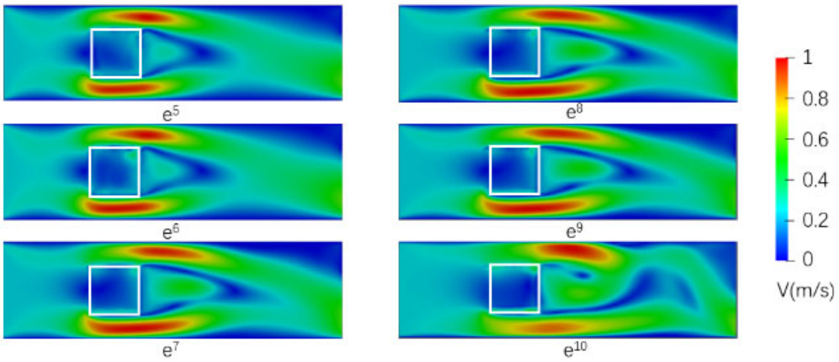

Figure 9. Velocity cloud maps of cross-sections inside pipes under different Rayleigh numbers

Figure 9 illustrates the flow state of the fluid at different Rayleigh numbers. When the Rayleigh number is less than e10, the state of the fluid is laminar. No matter how the Rayleigh number changes, the velocity and direction of the fluid do not change. When the Rayleigh number reaches e10, the state of the fluid is turbulent. When the fluid flows through the vibrating body, it creates a vortex at the end.

5. Conclusion

(1) As an open-source CFD software, OpenFOAM has a relatively complete set of functions, which can fully simulate fluid-structure interaction under various conditions, and provides more accurate simulation data. As an open-source CFD software, OpenFOAM has a relatively complete set of functions, which can fully simulate fluid-structure interaction under various conditions, and provides more accurate simulation data.

(2) Up-down vibration is helpful for heat transfer. To a certain extent, the flow path of the fluid is changed, as well as the flow velocity of the local fluid. The increase of wall temperature is significant for the improvement of heat exchange efficiency, and the higher the wall temperature, the effect of the average Nusselt number on the wall is also drastic.

(3) The shape of the vibrating body affects the heat exchange to a certain extent, but because the pipe simulated in this test is only 3m, the Nusselt number difference between the different shape of the vibrating body is not large. Perhaps larger in-pipe simulations will produce different results.

(4) The Rayleigh number has little effect on the wall Nusselt number when the fluid state is laminar, but when the Rayleigh number is greater than e10, the fluid state transitions to turbulent flow, and the Nu number will be greatly improved at this time.

References

[1] C.Z. Man, J.N. Hu, S.X. Yu. Numerical simulation of the enhanced heat transfer performance of a new twisted belt in laminar flow [J]. Thermal Science and Technology, 2019, 18(01): 42-49. (In Chinese)

[2] J.C. Han, L.R. Glicksman, W.M. Rohsenow, An investigation of heat transfer and friction for rib-roughened surfaces, Int. J. Heat Mass Transfer 21 (1978) 1143–1156.

[3] D.L. Gee, R.L. Webb, Forced convection heat transfer in helically rib-roughened tubes, Int. J. Heat Mass Transfer 23 (1980) 1127–1136.

[4] J.X. Zhu, M. Fiebig, N.K. Mitra, Numerical investigation of turbulent flows and heat transfer in a rib-roughened channel with longitudinal vortex generators, Int. J. Heat Mass Transfer 38 (1995) 495–501.

[5] Glowinski R, Pan T-W, Hesla T. A distributed lagrange multiplier/fictitious domain method for particulate fows. Int J Multiphase Flow 1999;25: 755–94.

[6] Cheny Y, Botella O. Set method for the computation of incompressible viscous flows in complex moving geometries with good conservation properties. J Comp Phys 2010;229: 1043–76.

[7] Peskin CS. Flow patterns around heart valves: a numerical method. J Comp Phys 1972;10(2):252–71.

[8] Minguez M, Pasquetti R, Serre E. High-order large-eddy simulation of flow over the ahmed body car model. Phys Fluids 2008;20.

[9] Isoardi L, Chiavassa G, Ciraolo G. Penalization modeling of a limiter in the tokamak edge plasma. J Comp Phys 2010;229: 2220–35.

[10] Constant E, Favier J, Meldi M, et al. An immersed boundary method in OpenFOAM: verification and validation[J]. Computers & Fluids, 2017, 157: 55-72.

[11] C.S. Peskin, Flow patterns around heart valves: a numerical method, J. Comput. Phys. 10 (1972) 252–271.