Heat transfer analysis of the bent cooling channels in SSRF light- blocked components

JIN Jianfeng[1,2] CHEN Haibo[1,*] XIAO Weiling[1] CHENG Wenlong[3] WANG Naxiu[4]

[1]*Department of Modern Mechanics, University of Science and Technology of China, Hefei 230027, China*

[2]*Beijing Institute of Space Long March Vehicle, Beijing 100076, China*

[3]*Department of Thermal Science and Energy Engineering, University of Science and Technology of China, Hefei 230027, China*

[4]*Shanghai Institute of Applied Physics, Chinese Academy of Sciences, Jiading Campus, Shanghai 201800, China*

Abstract The front end light-blocked components of the third generation of synchrotron radiation facility, which are subjected to high heat load, are cooled with flowing water through the cooling channels. The convective heat transfer coefficient and the flow resistance (or pressure drop) are two important parameters for evaluating the heat transfer performance of the cooling channels and should be strictly quantified. In this research, two typical bent cooling channels in Shanghai Synchrotron Radiation Facility (SSRF) were modeled and their in-pipe turbulent flows were simulated. The two criteria obtained under different channel water velocities met the SSRF technical requirements. To reduce the total pressure drop, arc transitions were proposed to replace the right angle transitions in the cooling channels. At the same time, an experiment was performed to measure the convective heat transfer coefficient of a typical bent channel unit. The experimental results were in good agreement with the simulation ones.

Key words Convective heat transfer coefficient, Pressure drop, Numerical simulation, Experimental test

1 Introduction

As a third generation synchrotron radiation facility, SSRF (Shanghai Synchrotron Radiation Facility) provides high brilliance and photon flux. The front end light-blocked components are required to withstand extremely high power and power density. The energy of the high heat load finally goes to the circulating cooling system through heat transfer of the cooling channels. The heat transfer coefficient of the cooling channel is one of the most important parameters when designing the front end components, as it directly affects the distributions of temperature and thermal stress of the components. On the other hand, the water flow in the cooling channels is driven by a pump. Thus the flow resistance character under different flow velocities is also a significant parameter and it also relates to the vibration status of the system.

So far, the key synchrotron radiation facilities in the world have completed a number of researches to enhance the heat transfer capabilities of the cooling channels. At Advanced Photon Sources (APS) in USA, the copper mesh was initially employed as porous insert to enhance the heat transfer in the cooling channels[1]. However, the copper mesh produced contaminants which blocked the channels in a certain degree and it was difficult to clear the contaminants. Since 2002, wire-coil inserts have been used as porous inserts and the heat transfer effect has improved significantly[2,3]. The Japanese third generation synchrotron radiation facility, SPring-8, was the first one in the world where the wire-coil inserts were used to increase the heat transfer ability of the cooling channels[4]. For smooth, grooved and wire-coil inserted tubes, a number of experiments were performed to measure the relationship between the heat transfer coefficient and the wire-coil diameter and thread pitch. The test results showed that the wire-coil

Supported by Natural Science Foundation of China (10975130).

* Corresponding author. *E-mail address*: hbchen@ustc.edu.cn

Received date: 2012-03-10

inserted tubes were capable to increase the heat transfer coefficient when the wire-coil diameter and thread pitch were properly chosen. But no obvious improvement was found for grooved tubes. At APS, Collins *et al.*[5] had performed more quantified experimental study. For 9.5 mm inner diameter cooling tubes, when the wire-coil diameter and the thread pitch were respectively 2.4 mm and 5 mm, the heat transfer coefficient reached 2×10^4 W·m^{-2}·K^{-1} under a flow velocity of 6 L·min^{-1}.

In the cooling channels of SSRF front end components, the porous inserts were not used as without them the heat transfer capability could satisfy the facility requirements. That is when the pressure drop is 2×10^5 Pa, the heat transfer coefficient is no less than 1.5×10^4 W·m^{-2}·K^{-1}. On the other hand, it is difficult to put any inserts in the long bent tubes and the maintenance cost will be high[6]. In this paper, the heat transfer character of SSRF bent tubes without porous inserts were studied through numerical simulation and experimental test. The commercial software MSC Fluent was used to determine the heat transfer coefficient and the flow resistance of Mask 1 and Mask 2 cooling channels under different flow velocities. An experimental system for measuring the heat transfer coefficients of the cooling channels was set up and experiments were performed to prove the reliability of the simulation results.

At last, to reduce the total pressure drops, arc transitions were proposed to replace the right angle transitions in the cooling channels. It would be an optimal alternative in the forth-coming design of the SSRF cooling channels.

2 Simulation Model

Figure 1 shows the configurations of Mask 1 and Mask 2 cooling channels, which belong to the SSRF front end light-blocked components of Wiggler and Undulator beam line, respectively. In geometric view, the cooling channel is periodic. So a single periodic part will be used for the next simulation. Fig.2 shows a single periodic part of Mask 2 cooling channel which contains two horizontal tubes, one vertical tube and one semicircular tube. The convective heat transfer coefficient and the pressure drop of this bent tube can not be evaluated from existing empirical formula. Therefore, our numerical simulation and experimental test were applied to predict these two key parameters. Mask 1 and Mask 2 cooling channels have some differences in dimensions and periodic numbers (Table 1). The fluid region in the single periodic cooling channel was discretized with the hexahedral finite element, with a total of around 400 thousand elements.

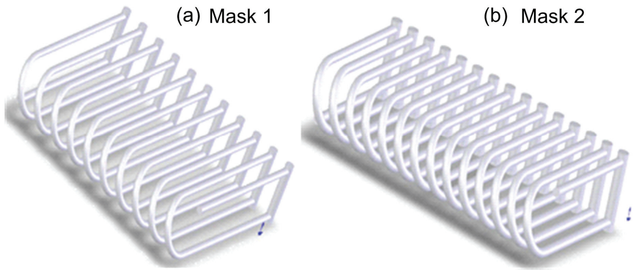


Fig.1 Configurations of the cooling channels in the two components.

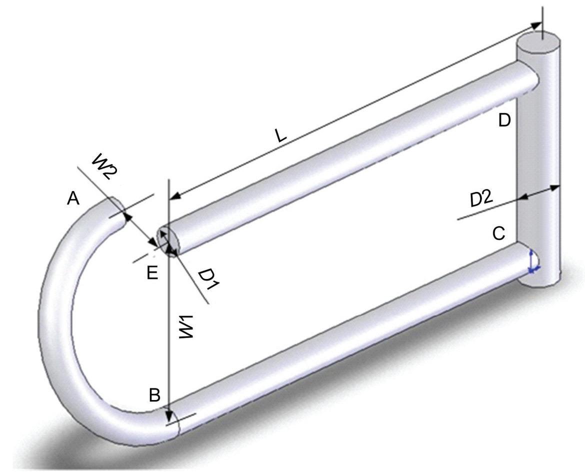


Fig.2 Single periodic model of Mask 2 in numerical simulation.

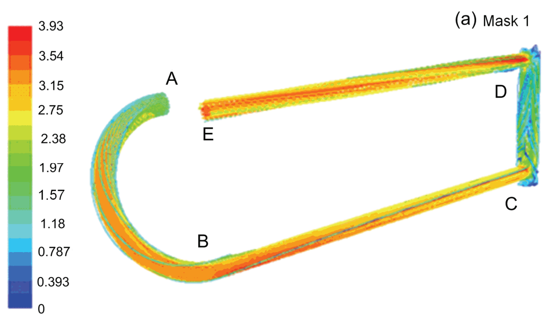
Table 1 Parameters of the cooling channels.

Component	$D1$ / mm	$D2$ / mm	$W1$ / mm	$W2$ / mm	L / mm	N
Mask 1	6	8	42	20	140	11
Mask 2	6	8	38	14	90	15

The flow velocities at the entrance of the cooling channel were controlled within 1–5 m/s, with respect to Reynolds numbers from 7 000 to 38 000, therefore the flow was turbulent in the channel. The flow non-uniformity at the entrance was ignored. The flow field and the heat transfer state were both assumed to be sufficiently developed. The standard $k-\varepsilon$ model was used for simulating the turbulent flow. The flow near the tube wall was dealt with a standard two-layer function model. The fluid in the channel was water which was idealized to be incompressible. For the boundary conditions, the flow velocities at the entrance were within 1–5 m/s and the temperature was 303.15 K. The relative pressure at the exit was set to 0 Pa. The whole tube wall was heated by a constant heat flux density of q_s=5.5×10^4 W·m^{-2}.

3 Simulation of the Heat Transfer Coefficient and the Flow Resistance

The single periodic model introduced in the previous section was imported in the software MSC Fluent for numerical simulation. The SIMPLE algorithm was adopted in the solution coupling of pressure and speed, while QUICK format was used in the solution of energy and momentum equations. Three dimensional flow patterns in Mask 1 and Mask 2 cooling channels were shown in Fig.3. It could be seen that the flow patterns were uneven due to the bent configurations, although the flow patterns of these two tubes were similar. In AB bent subsection, there was a rotational flow around the central line of the tube. In BC straight subsection, the flow turned to be more stable due to the smooth part and the flow resistance. However the flow was much more turbulent, because the CD subsection was composed of two right angle transitions where the tube contour changed sharply. Considering the mechanisms of convection heat transfer, rotational flow could promote the heat transfer ability and accordingly enhance the convective heat transfer coefficients. Under the same heat flux density, the larger the convective heat transfer coefficient was, the lower the wall temperature was. Fig.4 shows the wall temperature distributions of the two tubes. In the regions around C and D, the wall temperature varied significantly which implied a stronger turbulent flow.



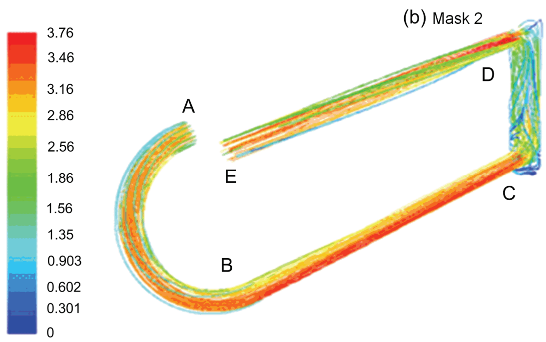
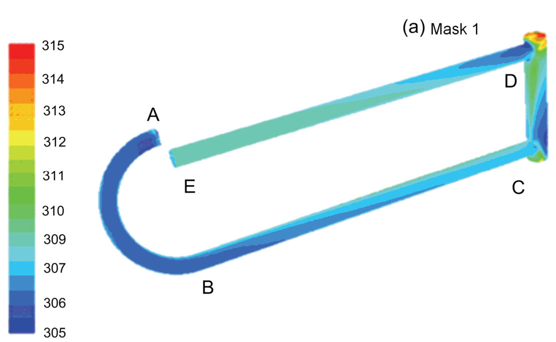


Fig. 3 Flow patterns of the two cooling channels.



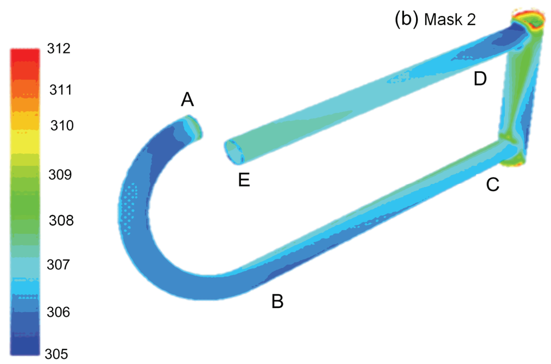


Fig.4 Wall temperature distributions of the two cooling channels.

As an important parameter, the convective heat transfer coefficient should be precisely quantified when designing the light-blocked components. Strictly speaking, there is a different local convective heat transfer coefficient at any point on the inner wall of the cooling channel. In practice, a stable local convective heat transfer coefficient is usually unavailable. Therefore, for such bent channels in

SSRF front end light-blocked components, we use the average convective heat transfer coefficient to describe the heat transfer ability, that is

$$h = \frac{q_s}{T_{w0} - T_{m0}} \quad (1)$$

where h is the average convective heat transfer coefficient, q_s is the constant heat flux density which is 5.5×10^4 W·m^{-2} in this research (evaluated from the experimental data), $T_{w0}=\int_S(T_w)\cdot dS/S_0$ is the weighted average temperature of the whole tube wall, and $T_{m0}=\int_V(T)\cdot dV/V_0$ is the volume weighted average temperature for the cooling water. T_{w0} and T_{m0} can be obtained directly from the simulation results post-processing. According to Eq.(1), the average convective heat transfer coefficients of Mask 1 and Mask 2 cooling channels under different flow velocities could be calculated. As shown in Fig.5, the coefficients of these two channels were almost the same and the relationship between the coefficient and the flow velocity was almost linear. The higher the flow velocity was, the larger the coefficient was. The design requirement of SSRF was that the coefficient should be no less than 1.5×10^4 W·m^{-2}·K^{-1}. This can be met when the velocity of cooling water reached 2.6 m/s, as shown in Fig.5 (the intersected point of vertical and horizontal dotted lines).

As another important parameter, the flow resistance is usually represented by the pressure drop which is defined to overcome the flow resistance in the tube[7], and it will finally determine the driving pressure of the pump. For a certain convective heat transfer coefficient, the flow resistance is the less the better. In each SSRF light-blocked components, the water in the cooling channel was pumped to a certain velocity to carry out the excess heat energy in time. The requirement of the pressure drop was no more than 2×10^5 Pa when the convective heat transfer coefficient was no less than 5.5×10^4 W·m^{-2}·K^{-1}. In the present simulation, the pressure drop for a single cycle channel was the difference between the entry and the exit pressures, multiplying the cycle number one would get the total pressure drop of the whole bent channel. Fig.6 shows the relationship between the total pressure drop and the flow velocity. The total pressure drops of these two channels were quite close to each other for all flow velocities. It was about 1.35×10^5 Pa at flow velocity of 2.6 m/s (the intersection point of vertical and horizontal dotted lines), and was less than the SSRF design requirement of 2×10^5 Pa (Fig.6).

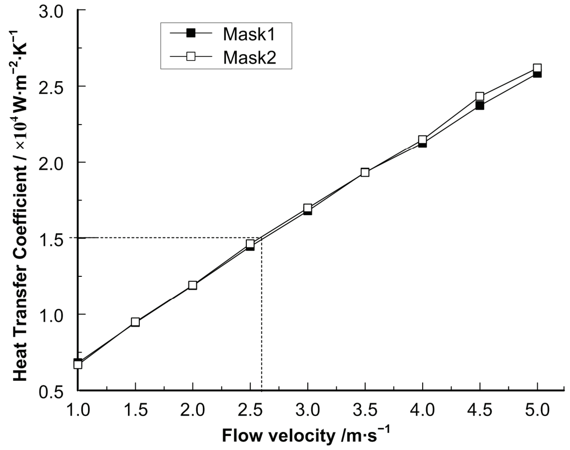


Fig.5 Relationship between the average convective heat transfer coefficient and the flow velocity.

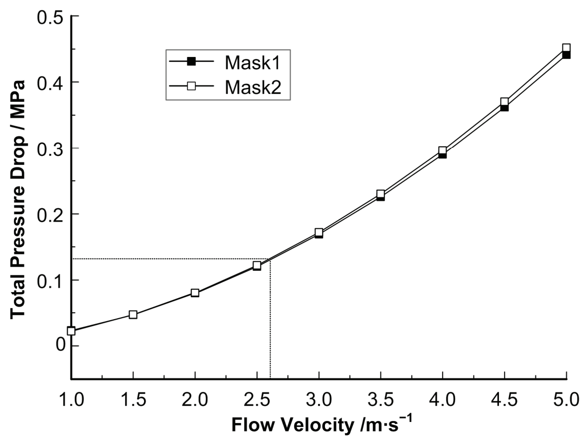


Fig.6 Relationship between the total pressure drop and the flow velocity.

4 Optimal Design of Cooling Channel

According to the heat transfer mechanisms, compared with straight tube, the bent tube can enhance the heat convection ability, but it will bring larger flow resistance. Especially for a bent tube with abruptly changing contour subsection, its local flow resistance will increase rapidly due to the strong turbulent flow. The tube subsection with right angle transition in SSRF belongs to this case. For Mask 1 with flow velocity 3 m/s in the cooling channel as example (Table 2), the total pressure drop was 15.378 kPa (sum of values listed in the first row), while the value at the right angle subsection CD was as large as 7.884 kPa which was over 50% of the total pressure drop. This concentration phenomenon was more obvious for the pressure drop per unit length, as listed in second row. The value at CD subsection was 150.846 kPa·m^{-1}, which was 5–8 times of those of other three subsections. This put us in mind that the configuration of the bent tube could be modified to reduce the pressure drop of CD subsection and finally to reduce the total pressure drop.

Table 2 Pressure drop and pressure drop per unit length of the subsections in Mask 1.

Channel Subsection	AB	BC	CD	DE
Pressure Drop / kPa	2.279	2.801	7.884	2.454
Pressure Drop Per Unit Length / kPa·m^{-1}	31.184	20.751	150.846	18.176

In fact, when the cooling water flows through the two right angle transitions, the flow state will change significantly which brings two obvious unfavorable factors for the cooling system and the light-blocked components[8]. Firstly, the flow resistance at the right angle transition is over half of the total pressure drop which finally significantly increases the flow resistance of the whole tube (Table 2). Secondly, vibration of the cooling channel is in close relation to the pressure pulse of water within the channel. The fluid pressure and flow velocity drastically vary at both right angle transitions, which may become the vibration sources in the tube system. However, the cooling channels with right angle transitions were produced in the first-stage seven beam lines of SSRF, after considering the two factors: (1) the drilling-connection process is simple; (2) the welding process is the less the better. In view of decreasing the pressure drop, an arc connection (modified model) was proposed in this paper to replace the right angle one (original model) in the cooling channel of Mask 1, as shown in Fig.7. Numerical simulation results showed that, for a certain flow velocity, the convective heat transfer coefficients were quite close to each other for the original and modified models, but the total pressure drop of the modified model was only 60% of the original one, as shown in Figs.(8) and (9). On the other hand, for a certain pressure drop, say 2×10^5 Pa, the original Mask1 cooling channel has a flow velocity 3.27 m/s, a flux 5.6 L/min and a convective heat transfer coefficient 1.74×10^4 W·m^{-2}·K^{-1}; the newly designed modified one would be 4.26 m/s, 7.2 L·min^{-1} or 2.12×10^4 W·m^{-2}·K^{-1}. Thus the modified model could remarkably increase the heat transfer ability. Now, the seven beam lines in the first stage of SSRF have been successfully constructed and the related welding technique has also been well developed. The new design proposed in this paper would be an optimal alternative for the forth-coming new beam lines in SSRF, and it would be a good reference for the beam line construction of other third generation synchrotron radiation facilities.

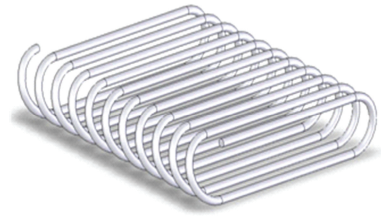


Fig.7 Modified model of Mask1 cooling channel.

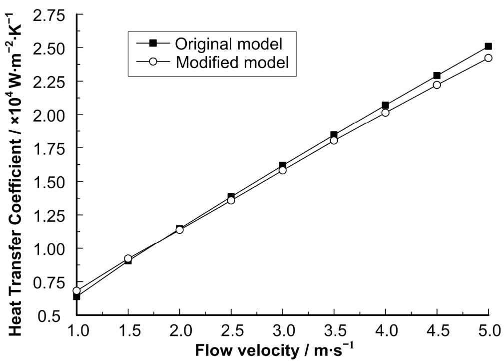


Fig.8 Comparison of convective heat transfer coefficients between original and modified models under different flow velocities.

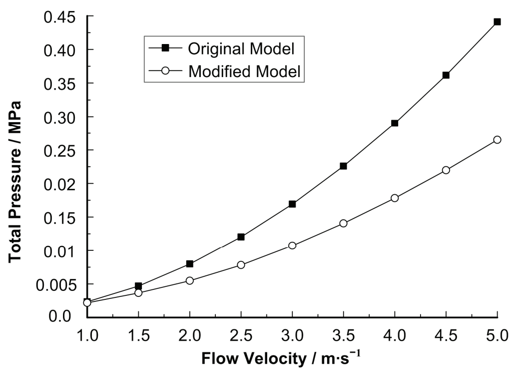


Fig.9 Comparison of total pressure drops between original and modified models under different flow velocities

5 Experiment for Heat Transfer Coefficient of Mask 1 Bent Channel Unit

To verify the accuracy of the above simulation results, an experimental system was designed and constructed to measure the convective heat transfer coefficient using the steady-state method[9]. According to the Newton's cooling law:

$$h = \frac{Q}{A(T_s - T_f)} \tag{2}$$

where h is the convective heat transfer coefficient of a tube with flowing water, Q is the heat flux through the tube wall, T_s and T_f are respectively the inner wall and the coolant temperatures, A is the internal surface area of the tube. The schematic diagram of the experimental set-up was shown in Fig.10.

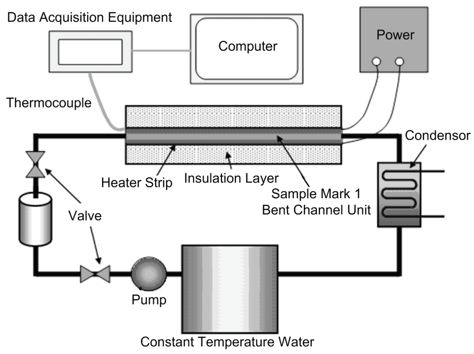


Fig.10 Schematic diagram of the experimental set-up.

Cooling water was pumped out by Pump from Constant Temperature Water Bath, and then it flowed into the test tube for heat exchange. The water eventually returned to the Water Bath after flowing through Condenser. The flow velocity was measured by a flow-meter. Temperatures for points on the tube inner wall and in the water were recorded once each 10 s via Data Acquisition Equipment by the T-type Thermocouples which were embedded near the tube wall and in the water, respectively. To eliminate the influence of flowing water, the thermocouples were packed in a stainless steel sleeves with a diameter of 1 mm. Several measuring points were set to reduce the random measurement error of the thermocouples. The test sample was heated by Heater Strip which was wrapped by Insulation Layer. Bent Channel Unit was the test sample. The water flow velocity was controlled within 1–5 m/s. Fig.11 is the photograph of the test sample which was wrapped by heat insulating materials.

When the experiment started, the temperatures on the tube wall and in the water rose from the room temperature at the same time. After some time, they both achieved the steady state and the temperatures measured by the thermocouples did not change any more. Arithmetic average was carried out to get the tube wall temperature T_s and the water temperature T_f. Then using the measured data of heating power, heat transfer coefficient of the test tube could be calculated through Eq.(2). Table 3 shows the experimental results of the Mask1 cooling channel unit under different flow velocities, as listed in first column. Compared with the numerical results, listed in the second column, a high agreement could be found, i.e. the deviations for all cases were no more than 8%. Thus it proved that previous simulation results were reliable.

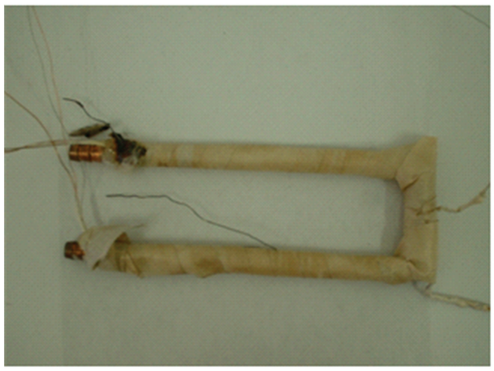


Fig.11 Photograph of the test sample.

Table 3 Comparison of the convective heat transfer coefficients between experiment and simulation for Mask 1 bent channel unit.

Velocity / m·s^{-1}	Experiment×10^4/W·m^{-2}·K^{-1}	Simulation×10^4/ W·m^{-2}·K^{-1}	Deviation / %
0	0.6334	0.5862	–7.4
2.0	0.9730	1.0263	5.5
2.5	1.2519	1.2460	–0.5
3.0	1.4059	1.4462	2.9
4.0	1.9138	1.8849	–1.5
4.7	2.1054	2.1137	0.4

6 Conclusions

In this research, numerical simulation and experimental test were carried out to study the heat transfer characteristics of the bent cooling channels in SSRF light-blocked components. The main conclusions are as follows. Within flow velocities of 1 –5 m/s, flow in the tubes are turbulent. The convection heat transfer coefficients and pressure drops under different flow velocities can be obtained through numerical simulation. The numerical results show that the heat transfer ability of the SSRF light-blocked component cooling channels met the requirements at present beam line operating conditions. If the right angle transitions are replaced by arc transitions in the cooling channels, the total pressure drop would greatly reduce, while the convective heat transfer coefficient almost remains unchanged. Thus, this is an optimal alternative design. As the convective heat transfer coefficients of Mask 1 cooling channel unit in the test are in good agreement with the numerical ones, it proves the correctness of the present numerical simulation for heat transfer of complex bent channels. In other words, when empirical formula is unavailable and experiment is difficult to perform, the numerical method for simulating heat transfer characteristics of complex cooling channels is practical and no loss of accuracy.

References

1 Kuzay T, Collins J, Koons J. Int J Heat Mass Tran, 1999, **42:** 1189–1204.

2 Jaski Y, Trakhtenberg E, Collins J. Thermo-mechanical analysis of high-heat-load components for the canted-undulator front end. 1st international workshop on mechanical engineering design of synchrotron radiation equipment and instrumentation, Wurenlingen, Switzerland, July, 2000.

3 Benson C, Trakhtenberg E, Jaski Y. Mechanical design of a front end for canted undulators at the advanced photon source. Eighth international conference on synchrotron radiation instrumentation, San Francisco, California, USA, August, 2003, 466–469.

4 Takiya T, Mochizuki T, Kitamura H. Development of enhanced heat transfer coolant channels for the spring-8 front end components. Spring-8 annual report, Harima Science Garden City, Hyogo, Japan, 1998, 164–166.

5 Collins J, Conley C, Attig J. Enhanced heat transfer using wire-coil inserts for high-heat-load applications. The 2nd international workshop on mechanical engineering design of synchrotron radiation equipment and instrumentation, Argonne, Illinois, USA, September, 2002.

6 Sharma S, Doose C, Rotela E. An evaluation of enhanced cooling techniques for high-heat-load absorbers. The 2nd international workshop on mechanical engineering design of synchrotron radiation equipment and instrumentation, Argonne, Illinois, USA, September, 2002.

7 Yang S M, Tao W Q. Heat transfer theory. Beijing, Higher Education Press, 2007, 162–165. (in Chinese)

8 Yuan J T, Tong L L, Cao X W. Nucl Sci Tech, 2010, **21:** 370–374.

9 Li Y, Yan C Q, Sun Z N, *et al.* Nucl Sci Tech, 2011, **22:** 122–128.