CFD simulations on the cooling effect of air supply velocity for high heat flux surfaces

W X Zhao, X Zhang*, X Y Xu, C Wu and W Ye
School of Mechanical and Energy Engineering, Tongji University, Shanghai, China
xuzhang@tongji.edu.cn

Abstract. During a process of building an experimental facility, the thermal bonding joint of organic glass bulk polymerization needs to be annealed by a heating belt with the heat flux density of 4200 W/m². The excessive heat needs to be removed efficiently to avoid damaging the surrounding areas. In this paper, the cooling effect of the air supply velocity on the high heat flux surface is investigated using CFD simulations, while the air supply temperature and volume are kept constant between cases. The results show that the air supply velocity has a significant impact on the cooling process of the high heat flux surface. With the increase of air supply velocity, the maximum and average temperature of the high heat flux surface decrease significantly. In addition, the cold air of large velocity could effectively suppress the spread range of hot air. These results provide a reference on the further study of organic glass annealing process.

1. Introduction
This research is based on a certain experimental facility with a large acrylic sphere whose inner diameter is 35.4 m, which is annealed and bonded by a layer of organic glass plates. During the annealing process of organic glass, an enormous amount of heat will be released and the thermal bonding joint’s temperature may reach hundreds of degrees. If the heat is not released in time, it would impede the process of annealing the organic glass and the safe operation of manufacturing personnel. It is necessary to utilize a certain cooling method to reduce the thermal bonding joint of organic glass - high heat flux surface’s temperature, and then control the indoor thermal environment to facilitate the craft process and ensure the operators’ safety.

The primary technical means that could cope with a challenge of cooling high heat flux surfaces are traditional cooling methods (air cooling, liquid cooling, heat pipe cooling, etc.), microchannel cooling, spray cooling, impact jet cooling, etc[1-3]. Air is a common medium used for cooling the high heat flux surface by natural convection or forced convection, because it is suitable for electronic devices and components of the heat flux density of less than 20W/cm². For the situation where other heat dissipation methods are restricted due to some practical conundrums, air cooling is the optimal cooling method[4,5]. At present, many scholars carried out researches on the air cooling effect on aerospace, electronic equipment and other fields with high heat dissipation requirement[6-9]. However, there are few research on the optimization of air supply parameters under the annealing process for organic glass. The optimal combination was mainly constituted by certain value of temperature, position, velocity, and angle of air supply, which reflected on thermal comfort of air-conditioned rooms and rational distribution of temperature and flow fields[10-15]. Due to the high surface temperature, its convective heat transfer with air and radiation heat transfer with other surfaces are different from those
in ordinary process plants and air-conditioned rooms, but the research techniques can be used to guide this research.

As for the high heat flux surface, there are some quandaries such as a huge amount of heat dissipation and the difficulty in cooling the hot surface. In general, it is hard to keep the air temperature steady in the experimental hall via conventional ventilation. It is necessary to adopt a local air supply and exhaust method, in other words, the inlet vents cold air from the upper part of the high heat flux surface and the hot air is quickly removed from the outlet at the lower part. And the control of inflow and outflow parameters is severely significant for surface cooling and local airflow organization. Above all, this paper focuses on the pure geometric parameters - the air supply velocity (that is the geometry of the air vent), while cooling the high heat flux spherical surface by forced convection. To optimize the design of the air supply and exhaust scheme and improve the energy efficiency, a series of simulations are carried out to investigate, which can guide for practical projects and future researches.

2. CFD methods

2.1. Physical model

The organic glass sphere with a diameter of 35.4m is assembled via mass polymerization by layers from top to bottom on the special mounting platform. The whole sphere is divided into 23 layers. When the layer is finished, the bonding seams need to be annealed by the heating belts. The organic glass sphere layered structure is shown in figure 1(a). For example, when the organic glass sphere is mounted to the equatorial layer, the equatorial layer is bonded to the upper sphere and then the heating belts are placed in the sphere inside and outside on the horizontal and vertical bonding seams with the width of 300mm~500mm. The inner and outer surface heating belts have the same power 10 kW per 6 m length respectively, about 1.7 kW/m, which is equivalent to 4200 W/m², and the heating time is 108 hours. The heating belt is schematically shown in figure 1(b).

Figure 1. The organic glass sphere layered structure(a) and the heating belts position(b).

The organic glass sphere is placed in the cylindrical experimental hall with a diameter of 43.5m and a height of 44m during the installation process (shown in figure 2). The ventilation scheme adopts a circular flexible air duct to be placed in the upper part of the heating belt to supply air and the lower part to exhaust air. Due to the complication of the physical model and several impact factors, it cannot accurately reflect independent influence on the temperature of the high heat flux surface when changing the parameters of air supply. And the hot air flow near the high heat flux surface is quickly discharged, not mixed with the mass of air in the experimental hall, which pertains to local airflow organization. Therefore, the organic glass bonding heating belt - high heat flux surface is abstractly
simplified to a T-shaped high heat flux surface and circular flexible air duct to a slot inlet/outlet. The simplified unit for studying is shown in figure 3.

![The organic glass sphere layout in the experimental hall.](image)

**Figure 2.** The organic glass sphere layout in the experimental hall.

![The simplified unit.](image)

**Figure 3.** The simplified unit.

The unit model is a rectangular parallelepiped with a cross-section of 1m×1m and the T-shaped heating belt is located on the side of the unit composed of two rectangles of 1m×0.4m. The top and the bottom surface of the unit are respectively installed the same slot with the length of 1m as the air inlet/outlet, and an uncertain width as a variable according to the air supply velocity.

The global grid size is designed as 0.04m and the air inlet/outlet are refined. The grid numbers of the calculation domain vary from 90 million to 100 million among designed conditions. Based on grid independence verification, the calculation result has little oscillation even through the grid is increasingly encrypted.

2.2. **Simulation content and methods**

The air supply temperature is 18℃, and the air supply volume of a simplified unit is calculated by the heat balance equation. The air supply velocity represents the amount of air supply per unit area, which has a significant impact on the jet flow and convective heat transfer coefficient. While other parameters of air supply keep constant, the velocity is designed as 1, 2, 3, 4, 5, 6, 7, 8 m/s by changing the width of slot inlet/outlet in order to investigate cooling effect of the air supply velocity on high heat flux surfaces.

The simulations are performed using ANSYS Fluent with Realizable k-ε turbulence model. Additionally, the enhanced wall function method is included. For the floating force caused by
temperature difference, the air density is calculated by Boussinesq hypothesis. The pressure velocity coupling adopts the SIMPLE algorithm, the pressure variable adopts the PRESTO! format, and the remaining variables adopt the Second Order Upwind format.

Table 1. Boundary conditions.

| Zone                                | Type          | Details                                                                 |
|-------------------------------------|---------------|-------------------------------------------------------------------------|
| The air supply vent                 | Velocity- inlet | Temperature is 18°C and velocity is set as 1, 2, 3, 4, 5, 6, 7, 8 m/s   |
| The air exhaust vent                | Pressure- outlet | The second type boundary condition as a constant heat flux of 4200 kW/m² |
| The high heat flux surface          | Wall          | The third type boundary condition of convection heat transfer            |
| The side surface where              | Wall          | Adiabatic boundary condition                                             |
| the high heat flux surface belongs  |               |                                                                         |
| The remaining surfaces              |               |                                                                         |

3. Results

3.1. Temperature distributions

As shown in figure 4. and figure 5, the surface temperature may be up to hundreds of degrees when the heat flux density is 4200 kW/m². The air supply velocity significantly influences the surface temperature when other parameters are constant. With the increment of air supply velocity, the maximum and average temperature of the high heat flux surface decrease gradually. Due to the air force convection, intensity of air blending increases along with the augment of velocity, which causes the larger ability of surface heat dissipation and lower surface temperature because of a bigger heat transfer coefficient.

Figure 4. High heat flux surface temperature distribution.
Figure 5. Curve of maximum and average temperature on the high heat flux surface as air supply velocity changes.

3.2. Flow distributions

Figure 6. The flow field streamline diagrams of the unit’s central section.
Due to the symmetric geometry of the unit model and the similarity of boundary condition, the central cross-section is selected to reflect the distribution of the flow fields. From figure 6, the flow fields of a single unit are mainly determined by two factors: hot air flow rising from the T-shaped heating belt and the cold air jet flow from the inlet. When the air supply velocity increases, the cold air can effectively hamper the diffusion of hot air, which suppresses the rise of the hot air flow, and controls the temperature distribution inside the unit.

4. Discussion
There are a great number of technical processes involving high heat dissipation problems, which require high efficient ventilation scheme designed for high heat flux surfaces. In this paper, the distribution of temperature and flow fields of the high heat flux surface of specific value in the air supply velocity range of 1~8m/s are discussed by CFD simulation, while the cooling effects of air supply temperature and position on the high heat flux surface of varying value have not be studied, which will be of great importance to the airflow organization design and equipment selection.

5. Conclusions and implications
In this paper, CFD simulation is used to investigate the cooling effect of air supply velocity on the high heat flux surface through the single-factor analysis. The main conclusions obtained from this study are as follows:

(1) The air supply velocity has a significant influence on cooling the high heat flux surface.

(2) With the increment of air supply velocity, the maximum and average temperature of the high heat flux surface decrease significantly while presenting much more flat distribution of temperature and flow fields. Moreover, air supply velocity is preferentially considered to alter the performance according to the surface temperature requirement in practice.

(3) When the air supply velocity increases, the cold air jet flow can effectively block the diffusion of hot air and control the temperature inside the unit.

References
[1] Chen S, Zhu L, and Sun Y 2015 Research progress of cooling technologies for high heat flux density devices Fluid Machinery 43(05) 39-45
[2] Wang Y 2011 Investigation of thermoelectric cooling technology on high heat flux density electronic components University of South China
[3] Huang D. and Yang S 2006 Cooling technique for high flux electronic Fluid Machinery (09) 71-74
[4] Drofenik U, Stupar A, and Kolar J 2011 Analysis of theoretical limits of forced-air cooling using advanced composite materials with high thermal conductivities Components Packaging and Manufacturing Technology, IEEE Transactions 1(4) 528-535
[5] He G 2014 Numerical simulation and experimental study of endothermic reaction applied to high heat flux heat dissipation South China University of Technology
[6] Zhang L, Liu X, Bi Z, Ji T, and Huang X 2018 Reviews of heat dissipation technology with high heat flux for high-power laser system Environmental Technology 36(02) 82-88
[7] Shi J 2016 Research and application of high intensity uniformity cooling technology after rolling for hot rolled strip Northeastern University
[8] Tao Y 2010 Study on the cooling of high-power solid-state lasers Graduate School of Chinese Academy of Sciences
[9] Liu M 2010 Numerical simulation of heat sink with combined micro channels and jet arrays for electric bus based on air supply parameters Automotive Practical Technology (16) 112-115
[10] Zhou K, Hu G, and Lang X 2017 Analysis of velocity field and temperature field in pure heat flux heat dissipation South China University of Technology
[11] Chen X, Shi J, Huan Z and etc 2017 Comparison of air-conditioning systems with bottom-supply and side-supply modes in a typical office room Applied Energy 07 78
[12] Wu F, Jun W, and Lin L 2015 The impact of split air condition supply temperature to indoor temperature field in winter Procedia Engineering 121 1689-1696
[13]  Hang Y, Ze Q, and Zhen J 2015 Numerical simulation of indoor thermal environment effected by air supply temperature and grille angle on stratum ventilation in a typical office Procedia Engineering 121 779-784

[14]  Zhao M, Luo X, Kong Q and Yu B 2005 Simulation and analysis of influence of inlet parameters on temperature gradient of floor air distribution system Refrigeration and Air Conditioning (06) 39-42

[15]  Zhou Y, Yu B, and Zhao R 2004 The numerical simulation and experimental study of air-conditioner inlet parameters’ influence on the temperature distribution and fluid flow inside room Refrigeration and Air Conditioning (03) 36-41