

Optimization on segmentation spraying for cooling a moving source at high temperature in a confined space

Chengqiang Zhi¹, Wei Ye^{1,2,*}, Qianru Zhang¹, Yixiang Huang¹, and Xu Zhang^{1,*}

¹School of Mechanical Engineering, Tongji University, Shanghai, PR China

²Key Laboratory of Performance Evolution and Control for Engineering Structures of Ministry of Education, Tongji University, Shanghai, PR China

Abstract. Heavy trucks, train carriages, and other moving vehicles with heating sources, are often loaded into a confined space for parking and maintenance. When the heating source from the vehicle is at high temperature, spray cooling using nozzles can be applied for at least two purposes. First, to further decrease the temperature of the surrounding areas. Second, to flush the surfaces of the vehicle as a primary cleansing method. To reduce water consumption, the number and positions of the running nozzles along the way need to be adjusted and optimized according to the position of the moving vehicle. To achieve that, the optimization of a segmented control method for the nozzles is discussed using both theoretical and computational fluid dynamics (CFD) methods in this paper. 120 ~ 240 nozzles in total were uniformly and linearly distributed on the ceiling of a 120m-long narrow space. The space between two adjacent nozzles are set at 0.5m, 0.7m and 1.0m, respectively. All the nozzles are divided into one to ten groups for segmentation control. One 35m-long vehicle with a heating source at over 800 K was moving at 0.05 m/s to 0.20 m/s. The results showed that, first, for one spraying group, at least 39 running nozzles were needed to minimize the areas of interior structure at temperature > 350K. Second, the water consumption can be reduced dramatically by increasing the number of groups. However, when dividing into more than six groups, the capacity of water saving is no longer significant. Third, when the vehicle is entering, multiple-group of running nozzles are needed to overcome the sudden heating source at high temperature.

1 Introduction

A confined narrow space is often seen in practice, there is a large turbine engine inside, exhausting a high-temperature jet at ~820 K and exit speed ~78 m/s. The exhaust gas needs to be cooled down to a safe level before discharging via the ventilation system. In practice, both dry and wet systems, i.e., spray cooling, could be applied to remove the heat of exhaust gas [1, 2]. Dry cooling system typically required a large-sized area to occupy, so dry cooling systems may not be applicable if space is limited or the heat sources are potentially moving. Wet systems can be more feasible and the cooling effectiveness of evaporative cooling is generally higher than direct dry air cooling [3-7], so wet systems are more appropriate for this occasion. However, wet systems require large amount of purified water. Therefore, it is worth studying how to reduce water consumption while meeting the cooling demand.

When the heating source is turned on at a fixed position, the high-temperature area (above 350 K) is fixed. Only cooling down this high-temperature area will meet the cooling demand. Turning on the nozzles in the low-temperature area is undesired and will bring invalid water consumption. When the heating source moves

from the interior of the cabin to the exit at a certain speed, the high-temperature area will spread as well. However, the expansion rule of the high-temperature area is not very clear at present. In the actual project, when the heating source moves, the nozzles installed on the ceiling are all turned on for spray cooling. When the heating source has not moved to the front, that is, the high-temperature area has not spread to the front. There is no need to spray at the front area, where activating the spray will result in relatively large invalid water consumption. So we came up with the idea of a segmented control method of nozzles activation. Divided the nozzles into groups and turning on the nozzles according to the position of heating source. Discussed whether this method had water saving capacity, i.e., ability to reduce the invalid water consumption. Therefore, the work of this paper has two parts. Firstly, the occasion when the impinging heating source at a fixed position without activating the nozzles was simulated by the CFD technology. Based on the relationship between the position of the heating source and the area of the high-temperature, the relationship between the moving speed of the heating source and the expansion speed of the high-temperature region was studied. Then, the invalid water consumption of 150

* Corresponding author: weiye@tongji.edu.cn, xuzhang@tongji.edu.cn

cases were calculated theoretically, when the heating source moves from the inside to the exit, with different heating source moving speed, the nozzles spacing, and the nozzles groups.

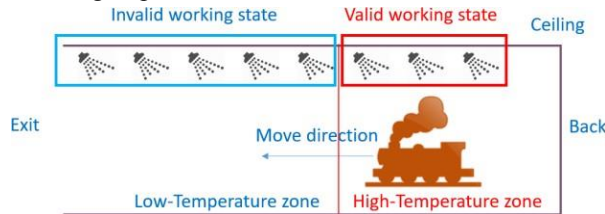


Fig. 1. Schematic diagram for the physical process.

2 Methodology

2.1 Numerical simulation

The relationship between the location of the heating source and the area of high-temperature was studied by the CFD technology. The actual length of the narrow space was $\sim 120\text{m}$, the width was $\sim 32\text{m}$, and the height was $\sim 10\text{m}$. There was an impinging heating source near the front of the narrow space and the rest of the space was empty. According to previous study, it had no influence on the front of space when the heating source lied in the rear of the space. Therefore, the length was reduced in the simulation, and only a 70m model was established. The heating source and the narrow space were bilaterally symmetric. In order to simplify the simulation, only half of the models were built in the CFD software, as shown in Figure 2. The ceiling, the back side, and the left side of the narrow space were wall surfaces, the right side was a symmetrical plane, and the front side was the exit. There were several axial fans on the ceiling near the back of the narrow space. The direction of the high-temperature jet was inclined 30 degrees to the upper left, with the height of 6 m , the outlet diameter of 0.95 m , the flow rate of $\sim 78\text{ m/s}$, and the temperature of $\sim 820\text{ K}$.

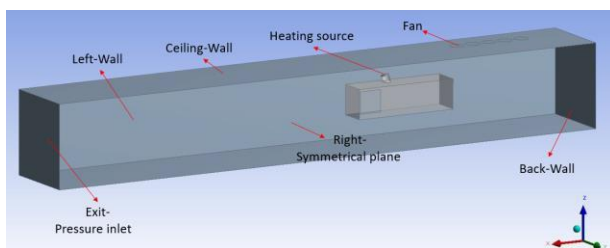


Fig. 2. CFD model of the space with impinging heating source.

For meshing, the boundary layer is divided and refined to avoid a velocity gradient at the ceiling and side wall. All the models adopted the shaped grid, and the total cells number was 3.9 million (with independence check), with local encryption in the heating source exit area and the ceiling area. The $k-\varepsilon$ realizable turbulence model and the standard wall function was selected in this study. The coupling between the fluid speed and the pressure was calculated via the SIMPLE algorithm. The momentum, turbulence energy, dissipation rate of turbulence, and the

discretization of Reynolds stress are used to adopt a second-order up-wind scheme, while the discretization of the pressure adopts the standard form. The study used transient simulation with a time step of 1 s .

2.2 Invalid water consumption calculation method

The heating source moved linearly from the back to the exit along the length direction, that was, along the x -direction in Figure 2. The nozzles were arranged on the ceiling along the x -direction. When a nozzle was turned on to spray at the high-temperature area, the nozzle was in valid working state. When a nozzle was turned on but spray at the low-temperature area, the nozzle was in invalid working state and caused large invalid water consumption.

Nozzles were arranged on the ceiling along the x direction. The spacing of nozzles was $\delta\text{ m}$, so the total number of nozzles was N . When the heating source started to move, there were N_H nozzles in the initial high-temperature area and N_L nozzles in the initial low-temperature area. As the heating source moved, the high-temperature region expanded along with it, and the number of nozzles in invalid working state decreased. When high-temperature area expanded to the last nozzle, all the nozzles were in valid working state.

Assumed the high-temperature area moved at a speed of $a\text{ m/s}$. Every $t = \delta/a\text{ s}$, the high-temperature area passed through the length of the interval between two nozzles. The total time of the high-temperature area expanded to the last nozzle was $T = (N_L - 1)t\text{ s}$. Divided the total time into $N_L - 1$ stages. The amount of invalid water in each stage W_i can be calculated as follows:

$$W_i = N_i q t \quad (1)$$

where N_i is the number of nozzles in invalid working state in each time stage; q was the water flow rate of each nozzle.

The total amount of invalid water W was the sum of W_i in each stage.

If the nozzles were grouped too much, multiple groups of nozzles needed to activate at the initial time stage. This was due to only one group of nozzles was insufficient to cover all of the high-temperature area. The amount of invalid water of 150 working condition were calculated when changing the nozzles spacing 0.5 m , 0.7 m , 1.0 m , the heating source moving speed 0.056 m/s , 0.067 m/s , 0.083 m/s , 0.111 m/s , 0.167 m/s , and the groups that nozzles are divided into $1, 2, 3, 4, 5, 6, 7, 8, 9, 10$.

3 Results

According to the results of numerical simulation, the contour of temperature at different moments were shown in Figure 3. It was found that the temperature of flow field became stable at $\sim 8\text{ s}$, and the length of initial high-temperature area was $\sim 40\text{ m}$. So we made a reasonable assumption that the diffusion speed of the high

temperature area and the moving speed of heating source was the same when heating source move at low speed. Based on the assumption, the amount of wasted water of 150 cases were calculated when the heating source moved from inside to exit.

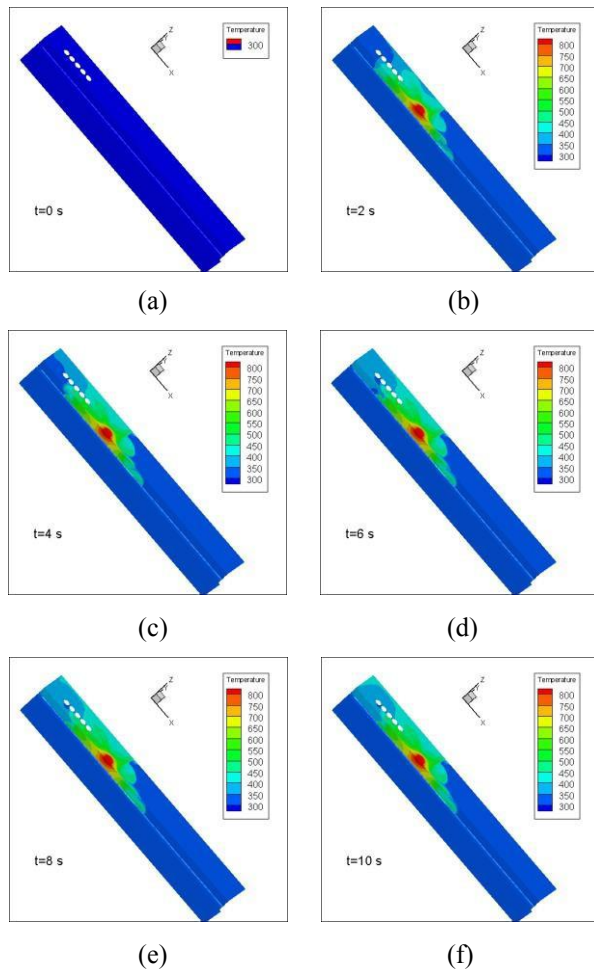
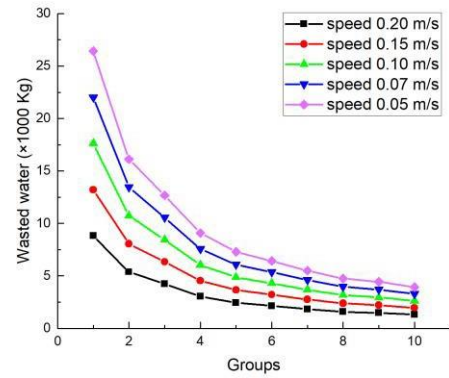


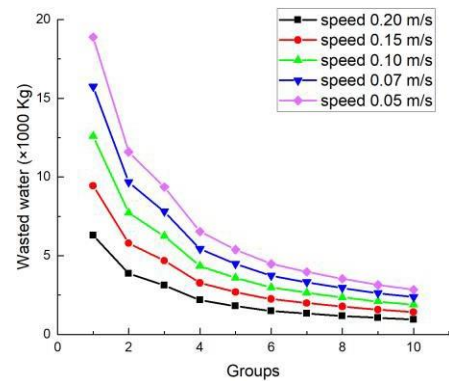
Fig. 3. Temperature contours of the ceiling and left side wall.

The invalid water of different groups of nozzles was shown in Figure 4(a-c). As can be seen, regardless of the spacing of nozzles, the invalid water decreased with increasing the moving speed of heating source at the same group, which was due to the invalid working time of nozzles was short. When compared Figure 4 (a), (b) and (c), it was found that the invalid water decreased with increasing the spacing of nozzles at the same group and the same moving speed. This was because the number of nozzles decreased when the spacing increased at a fixed length of ceiling.

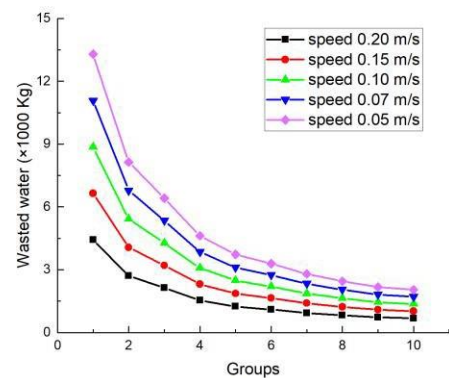
Furthermore, when all the nozzles were turned on simultaneously, i.e., the nozzles were divided into 1 group, a large amount of water was wasted. When groups of the nozzles were less than six, the water saving capacity increased dramatically with increasing groups of the nozzles, beyond this, the water saving capacity increased slightly. This is due to a decrease in the number of nozzles at per activation. Therefore, in order to simplify the system and reduce the cost, the nozzles divided into six groups was adaptive.



(a) spacing 0.5m



(b) spacing 0.7m



(c) spacing 1.0m

Fig. 4. Invalid water consumption in different working condition.

4 Conclusions and limitations

4.1 Conclusions

A segmented control method of nozzles was proposed. The relationship between the location of the heating source and the high-temperature area was studied by the CFD method. Assumed the expansion speed of the high-temperature area equal to the moving speed of the heating source reasonably. By changing the nozzles spacing, the heating source moving speed, and

the groups that nozzles are divided into, the invalid water consumption was calculated of 150 cases by changing the nozzles spacing, the heating source moving speed, and the groups that nozzles are divided into. Major conclusions are as follows:

(1) The temperature distribution of flow field became stable at ~8s, and the length of initial high-temperature area was ~40m. At least 40 nozzles are needed to decrease the temperature of high-temperature areas. If the nozzles were grouped too much, multiple groups of nozzles needed to activate at the initial time stage.

(2) The segmented control method of nozzles saved purified water effectively. When groups that the nozzles were divided into less than six, the invalid water consumption decreased dramatically with increasing groups of the nozzles, beyond this, the invalid water consumption decreased slightly. The segmented control method of nozzles has a large water saving capacity in spray cooling.

4.2 Limitations

More studies are needed to further understand the exactly relationship between the expansion speed of the high-temperature area and the moving speed of the heating source, and how to implement this control system in practice.

Acknowledgment

This research has been supported by the National Natural Science Foundation of China under Grant No. 51878463.

References

1. J.G. Bustamante, A.S. Rattner, S. Garimella, *Appl. Therm. Eng.*, **105**, 362 (2016).
2. Y.Q. Kong, W.J. Wang, X.W. Huang, L.J. Yang, X.Z. Du, Y.P. Yang, *Appl. Therm. Eng.*, **119**, 254 (2017).
3. N.Y. Zhou, F.J. Chen, Y.C. Cao, M.M. Chen, Y. Wang, *Appl. Therm. Eng.*, **112**, 1117 (2017).
4. Z.F. Zhou, R. Wang, B. Chen, T. Yang, G.X. Wang, *Appl. Therm. Eng.*, **102**, 813 (2016).
5. J.H. Liu, R. Xue, L. Chen, X.F. Liu, Y. Hou, *Exp. Therm. Fluid Sci.*, **75**, 89 (2016).
6. M.H. Sadafi, S.G. Ruiz, M.R. Vetrano, I. Jahn, J. van Beeck, J.M. Buchlin, K. Hooman, *Energy Conv. Manag.*, **108**, 336 (2016).
7. H. Bostanci, B. He, L.C. Chow, *Int. J. Heat Mass Transf.*, **107**, 45 (2017).