A Simplified Method of Radiator to Improve the Simulation Speed of Room Temperature Distribution

Zhenqiang Cao, Haiyi Sun, Tong Niu and Xia Lu

August 30, 2020

A simplified method of radiator to improve the simulation speed of room temperature distribution

Zhenqiang Cao[1], Haiyi Sun[2]* ,Tong Niu[1] and Xia Lu[3]
[1] School of Mechanical Engineering, Shenyang Jianzhu University, 110168, Shenyang, China
[2] College of Science, Shenyang JianZhu University, 110168, Shenyang, China
[3]School of Management, Shenyang Jianzhu University, 110168, Shenyang, China
*shy_xx@163.com

Abstract. Computational fluid dynamics (CFD) has become one of the important methods of indoor environmental analysis. Because the indoor environment of the building has the characteristics of large space and complex structure, the calculation amount of directly using CFD to simulate the indoor temperature distribution is so huge that the simulation efficiency is too low for practical application. Based on the shortcomings of the existing building environment simulation methods, Solidworks Flow Simulation and MATLAB are used to carry out research on indoor temperature distribution simulation methods. The overall treatment is equivalent to a constant heat source, which simplifies the heat dissipation model and avoids simulating complex flow patterns. Compared with traditional simulation methods, it can greatly improve simulation efficiency and model solving speed while ensuring simulation accuracy. The overall method is well supported by the simulation results.

Keywords: Computational Fluid Dynamics (CFD), Temperature distribution, Micro segment simplified, Solidworks Flow Simulation.

1 Introduction

How to predict the indoor environment of buildings is an important research topic in the field of HVAC (Heating, Ventilation and Air Conditioning). It starts from computational methods and uses the rapid computing power of computers to obtain approximate solutions to fluid control equations. Computational Fluid Dynamics (CFD) emerged in the 1960s. With the rapid development of computers after the 1990s, CFD has developed rapidly and has gradually become an important method of building environment simulation. Using a computer for CFD analysis can perform coupling simulations of multiple variables, and it can more clearly understand the complex process mechanism inside the system [1-5]. In the field of heat transfer simulation, CFD analysis is widely used for energy conversion of car radiators, computer radiators, fuel cells and nanotubes [6-10]. CFD analysis is also commonly used in the field of building environment simulation. By simulating different heat source systems, the indoor temperature distribution under different conditions can be obtained [11]. In order to meet the thermal simulation of complex shapes, clustering hexahedral elements can be used to use ray intersection and ray/triangle intersection gridding methods to ensure sufficient simulation accuracy [12]. In order to improve the simulation efficiency, the mathematical model of the simulation object can be simplified. For example, indoor temperature distribution, heating inlet and outlet temperature, and shape characteristics of the radiator are analyzed after simplified modeling [13-16]. For example, when the radiator is working, the indoor air will undergo natural convection in which the heated air rises along the wall and then cools down [17]. The use of "user-defined wall function" to simplify the modeling of air convection can improve simulation efficiency. Compared with the k-ω SST, (Shear Stress Transfer), turbulence model, this model greatly reduces the number of units and improves the simulation solution speed. However, this simplified method introduces a "user-defined wall function", which makes the modeling process more complicated [18].

According to the general simulation method, both hot water and hot air must be modeled and analyzed at the same time. Because the internal structure and flow field of the heating system are complicated, the calculation amount is too large, the simulation speed is slow, and the simulation may even fail. Therefore, we simplified it by the following method.

In sect.2, take a specific room as an example. According to the thermal differential equation and various boundary conditions and the heat dissipation capacity of pipes, walls and glass, the radiator is transformed into a constant heat source. Summing the equivalent heat dissipation power of each unit's pipeline, and its power can be regarded as the radiator power under the changed environment temperature.

When the heating power of the indoor heating is equal to the heat dissipation power, the corresponding ambient temperature is the desired equilibrium indoor temperature. In sect.3, input the established model into Solidworks Flow Simulation to find the temperature distribution and average temperature in the room. Compared with the simulation result of the traditional method, the simulation result shows that the error of the average temperature is less than 0.2%, which is completely acceptable for the indoor temperature, which verifies that the simplified method is feasible and reasonable. The simulation time was shortened from 1 hour and 26 minutes to 34 minutes, which improved the simulation efficiency.

2 Simplified mathematical model based on raditor discretization

2.1 Simplified mathematical model of radiator heat dissipation power

Generally, if CFD directly uses the input water temperature and flow rate as the heat source model, it takes a long time for the simulation software to build the heat transfer model. In the transient special conduction, the solution of the temperature field distribution over time may also get results that are inconsistent with common sense.

In order to optimize the calculation speed, considering the heating effect, it can be simplified as a constant heat source to achieve the purpose of simplifying the model. The internal structure of the coil radiator is an S-shaped coil with hot water flowing through it. The radiator can be converted into a circular heat pipe by using the principle of the same volume. Solving heat generation power using thermal differential equation

$$\begin{cases} V = L \times D \times H \\ V = \pi \times r^2 \times l \end{cases} \quad (1)$$

The V is radiator water capacity; L, D, H is radiator geometry; l is equivalent pipe length for heating; r is inner diameter of equivalent radiator pipe.

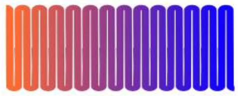

Fig. 1. Schematic diagram of simplified radiator structure

The following will take a standard room as an example. The specific parameters are shown in the **figure.2** below to verify the simplify method proposed in this article.

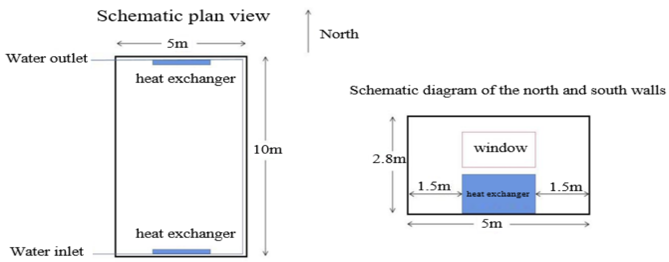

Fig. 2. Schematic diagram of the room

When the inlet flow rate is known, the length of the radiator pipe water per second is taken as a unit length pipeline, and the heat conduction differential equation and the third type of boundary conditions are used to solve the equivalent radiator pipe heat dissipation power.

According to the continuity equation of fluid mechanics:

$$q = Av \quad (2)$$

The A is pipe section area; v is pipeline water velocity.

$$l_{per} = \frac{q}{\pi \times r^2 \times 3600} \quad (3)$$

The l_{per} is the length of a micro segment.

The distance that the water flows per unit time is regarded as the length of the micro-segment. Each micro segment can be regarded as a round wall radiator conduction, as shown in the following figure:

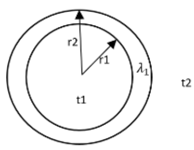

Fig. 3. Schematic diagram of iron pipe heat conduction

$$\begin{cases} Q_2 = 2\pi r_1 l h_w (t_{nb} - t_{gw}) \\ Q_2 = 2\pi r_2 l \frac{(t_{wb} - t_{nb})}{\ln(r_1/r_2)} \\ Q_2 = 2\pi r_1 l h_q (t_n - t_{wb}) \end{cases} \quad (4)$$

The Q_2 is micro segment heat exchanger; r_1 is pipe inner diameter; l is micro segment length; h_w is convective heat transfer coefficient of water; t_{nb} is pipe inner wall temperature; t_{gw} is water temperature in pipe; λ_2 is convection heat transfer coefficient of iron pipe; t_{wb} is pipe outer wall temperature; r_2 is outer diameter of pipe; h_q is air convection heat transfer coefficient.

According to Fourier's law of heat conduction [19] and the third type of boundary conditions, the heat exchanger balance equation is established. The heat dissipation of each micro-segment can be calculated by MATLAB, and the heat of each micro segment body can be added up. The total heat lost by the pipeline in 1 s is obtained, which is the heat generation power of the heating system.

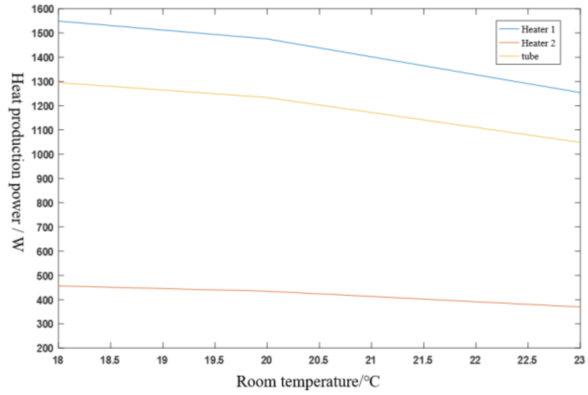

Fig. 4. Heat dissipation power diagram of each part of the pipeline

2.2 Simplified mathematical model of windows and walls heat dissipation power

When analyzing the room model, all the energy in the room is input through the radiator, and all the energy output is completed through the exterior wall and glass. When the output energy is balanced with the input energy, the system will reach a steady state, so a heat dissipation model can be established:

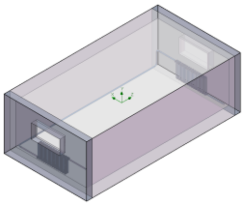

Fig. 5. 3D view of the room

It is known that the heat in the room is dissipated through the glass. In order to make the room warmer, double-layer glass is usually used in the house. However, for this article, the temperature distribution of the room is mainly considered. The coefficient is close to that of double-layer glass, which can be used in this study. The heat flux per unit time obtained by Fourier's law [19]:

$$Q_1 = \lambda_1 A \frac{t_2 - t_1}{d} \tag{5}$$

The Q_1 is heat flux density between inner and outer glass; λ_1 is thermal conductivity of glass; t_1 is surface temperature of glass in contact with outdoor; t_2 is Glass and indoor contact temperature; A is glass surface area.

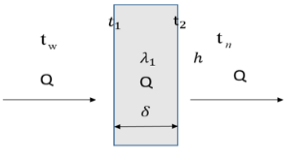

Fig.6. Schematic diagram of solid heat conduction

$$Q_2 = hA(t_w - t_1) \tag{6}$$

The Q_2 is heat flux density between environment and outer glass; h is thermal convection coefficient of air; t_w is outdoor temperature.

The heat exchange between indoor air and glass can be expressed by the following formula:

$$Q_3 = hA(t_2 - t_n) \tag{7}$$

The Q_2 is heat flux density between inner glass and indoor; t_2 is indoor side glass temperature t_n is indoor temperature.

Since the surface temperature of the glass in contact with the outdoor and indoor sides is unknown, but the ambient temperature of the outdoor and indoor is known. So the heat dissipation power can be solved using the air convection coefficient as shown in **Figure.7.8.**

The heat dissipation power of the concrete exterior wall at a room temperature of 21 degrees Celsius and an ambient temperature of 0 degrees Celsius is 685.02W.

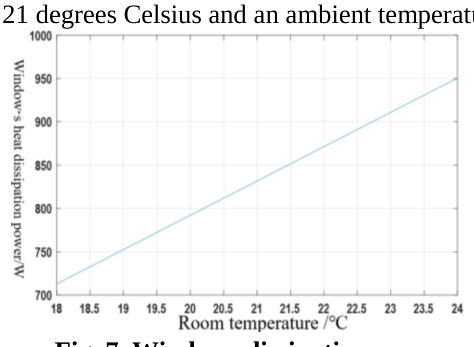

Fig. 7. Windows dissipation power

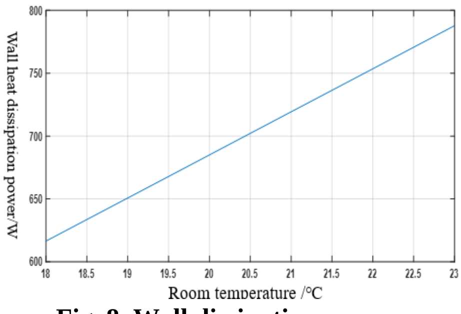

Fig. 8. Wall dissipation power

When the dissipated power is consistent with the heat generated power, the heat will reach a dynamic balance. As shown in **Fig. 9** the horizontal axis of the intersection is the power corresponding to the heating during thermal equilibrium, and the vertical axis is the average indoor temperature during thermal equilibrium.

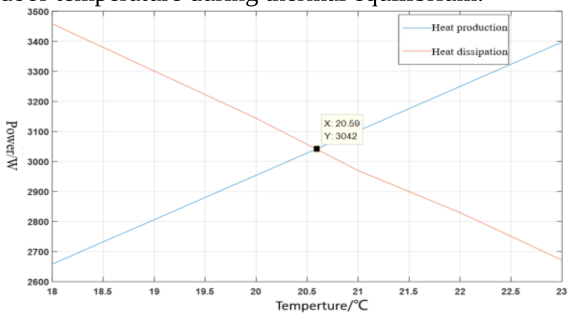

Fig. 9. System heat dissipation diagram

3 CFD Simulation

In order to improve the simulation efficiency and reduce the computer solution time, a simplified simulation method is used in this section, which is specifically as follows: treat the radiator and the internal hot water as a whole, and make it equivalent to a constant heat source. When using the heat dynamic balance obtained in **Fig. 9**, the heating power is used as the heat generation power of the Constant heat source.

Input the equivalent model into a computer, perform fluid simulation analysis, and solve the temperature distribution inside the room. At the same time, the average temperature of the room when the temperature is dynamically balanced is obtained through the simulation, and the accuracy of the heating power and the average indoor temperature corresponding to the heating power and the indoor average temperature when the heat is dynamically balanced is verified by the above method. The room temperature distribution obtained by the simulation is shown in the **Fig.10.11.12** Shown:

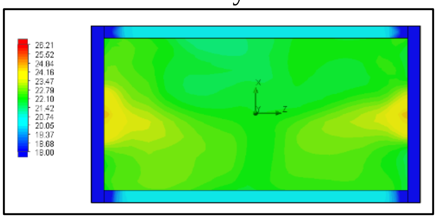

Fig. 10. Temperature distribution XZ plan

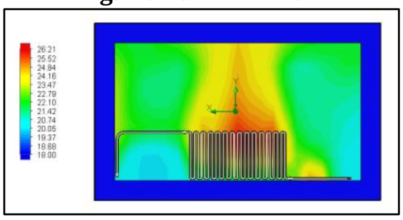

Fig. 11. Temperature distribution XY plan

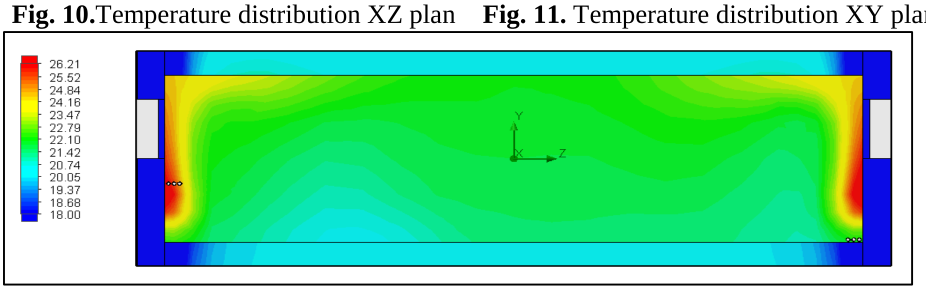

Fig. 12. Temperature distribution YZ plan

The average room temperature of the simulation is 20.1786°, which is 2% error from the simplified method above, so the simplified model above is feasible. The main reason for the error is the simplified heating of complicated shapes into long straight pipes. Compared with the real heating pipes, the heat convection situation is different and the equivalent heating power is higher. Through computer simulation, the real thermal convection flow line diagram can be obtained as shown in **Fig. 13**.

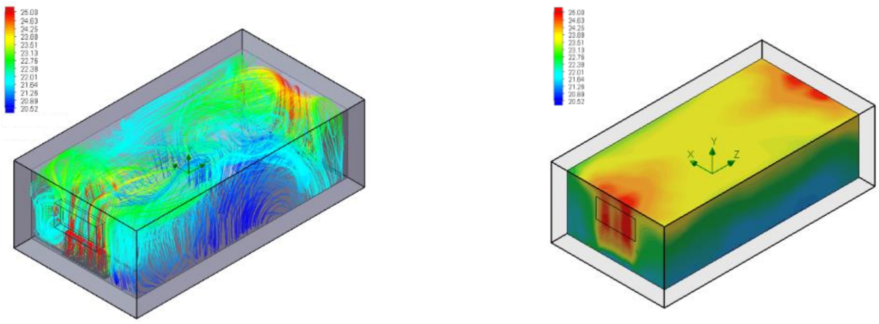

Fig. 13. Room temperature distribution map

4 Conclusion

An important reference for the reasonable design of HVAC （Heating, Ventilation and Air Conditioning）equipment is to simulate indoor plumbing equipment to optimize indoor temperature distribution and reduce energy waste. The main purpose of this paper is to propose a method to simplify the heating model of indoor plumbing equipment by avoiding the simulation of complex flow patterns, thereby increasing the solution speed and making large-scale simulation possible. Comparing the simplified simulation method with the traditional simulation effect, the same object can be simulated under the same device. The traditional method takes 1 hour and 26 minutes to solve, while the simplified method takes 34 minutes. The simulation efficiency has been significantly improved. It is effective and feasible to equate the temperature and flow rate of the hot water inlet with a constant heat source. Make the indoor heat equal to the heat dissipation power at different indoor and outdoor temperatures, and then adjust the inlet water temperature and flow rate, so that the indoor temperature is stable and comfortable, and more convenient.

Acknowledgment

This work was partially supported by Liaoning Provincial Department of Education Scientific Research Fund Project-Basic Research Project (2020018), Liaoning BaiQianWan Talents Program (grant no. 2017076) and the Natural Science Foundation of Liaoning Province (grant no. 20170540769).

References

1. Pinghai S., Ken D., Thom M.: Algae-dewatering using rotary drum vacuum filters: Process modeling, simulation and techno-economics. Chemical Engineering Journal 268,67-75(2015).
2. Shokufe A., Ali G., Nima R.: Mathematical modeling and simulation of water-alternating-gas (WAG) process by incorporating capillary pressure and hysteresis effects. Fuel, 263,111362 (2020).
3. Krzysztof B., Antoni R., Henryk W.: Thermal Analysis of the Factors Influencing Junction Temperature of LED Panel Sources. Energies 12(20), 3941 (2019).
4. Gustavo R., Luciano G., Mario S.: Numerical and experimental thermo-fluid dynamic analysis of a power transformer working in ONAN mode. Applied Thermal Engineering 112,1271-1280(2017).
5. R. Kirubagharan, C. Ramesh, P. Pragalathanl.: Geometrical analysis of automobile radiator using CFD. Materials Today: Proceedings (2020).
6. Pradeep P.: CFD Analysis of Helical Tube Automobile Radiator Considering Different Coolants. Industrial Engineering Letters 8(5),14-20 (2018).
7. Qinguo Z., Liangfei X., Jianqiu Li.: Performance prediction of plate-fin radiator for low temperature preheating system of proton exchange membrane fuel cells using CFD simulation. International Journal of Hydrogen Energy, S0360319917331208 (2017).
8. Chunhui Z., Mesbah U., Austin C.: Full vehicle CFD investigations on the influence of front-end configuration on radiator performance and cooling drag. Applied Thermal Engineering 130,1328-1340 (2018).
9. Luciano G., Gustavo R., Mario S.:. Reduced model for the thermo-fluid dynamic analysis of a power transformer radiator working in ONAF mode. Applied Thermal Engineering 124,855-864 (2017).
10. Ali K., Masoud A.: Numerical study on thermal performance of an air-cooled heat exchanger: Effects of hybrid nanofluid, pipe arrangement and cross section. Energy Conversion and Management 164,615-628 (2018).
11. Daniel R., Mikael R., Lars W.: CFD modelling of radiators in buildings with user-defined wall functions. Applied Thermal Engineering 94,266-273 (2016).
12. Yang S., Pilet T., Ordonez J.: Volume element model for 3D dynamic building thermal modeling and simulation[J]. Energy 148(APR.1),642-661 (2018).
13. Adnan P., Sture H.: Low-temperature ventilation pre-radiator in combination with conventional room radiators. Energy and Buildings 65,248-259 (2013).
14. Tamer C., Hakan O., Senol B.: Determination of the effects of different inlet-outlet locations and temperatures on PCCP panel radiator heat transfer and fluid flow characteristics. International Journal of Thermal Sciences 121,322-335 (2017).
15. Weizhen L., Andrew T., Alan P.:Prediction of airflow and temperature field in a room with convective heat source. Building and Environment 32(6),541-550 (1997).
16. Pooranachandran K., Vellaisamy K., Ramalingam V.: FANNING FRICTION (F) AND COLBURN (J) FACTORS OF A LOUVERED FIN AND FLAT TUBE COMPACT HEAT EXCHANGER. Thermal Science Journal 21(1A),141-150 (2016).
17. El-Gendi, Mahmoud.: Transient turbulent simulation of natural convection flows induced by a room radiator[J]. International Journal of Thermal Sciences 125,369-380 (2018).
18. Mohammed S., Essam E.:CFD Investigation of Air Flow Patterns and Thermal Comfort in a Room with Diverse Heating Systems. Current Environmental Engineering 6(2),150-158 (2019).
19. Yang S., Tao W.: Heat Transfer Third Edition[M]. Higher Education Press (1998).