A Fast Computational Fluid Dynamics Model for the Flow and Heat Transfer Characteristics Analysis of Indoor Substation Rooms

L. W. Xu * and A. X. Zou

Chongqing Electric Power Test and Research Institute, Chongqing 400015, China

710231199@qq.com

Abstract. In order to find a high efficiency, energy saving structure of the indoor substation room, a novel computational fluid dynamics method is proposed to model its flow and heat transfer characteristics. As an automatic optimization system, the efficient methods for mesh generation and numerical calculation are necessary. The mesh of the whole room including the transformer and its heat transfer fins is created using a structured mesh scheme, and a detection technology is adopted to recognize all the obstacles. In order to improve the numerical efficiency, a fast computational fluid dynamics (FFD) method is adopted to establish the three dimensional dynamic model with heat source. The comparisons between the FFD model and the experimental data show that the FFD can model the flow and heat transfer characteristics of the indoor substation efficiently. By modifying the related parameters in the model, this method can be used in the simulation of other indoor substation rooms to optimize their structure and operation.

1. Introduction

In recent years, with the acceleration of urbanization in China and the rapid growth of electricity demands, the urban power grid has been developing rapidly. In order to save the city land and reduce noise, indoor substation is increasingly widely used in the city power supply system [1-3]. However, the closed environment brings the big problems of ventilation and heat exchange, especially in the summer, it is difficult to release the heat generated by the transfers in the substation room. The high temperature will directly threat the safety of the transformers [4-7], and more than half of the failures of transformer are caused by its over-temperature [6]. In order to study the ventilation and heat transfer characteristics of indoor transformer substation room, and to seek effective means to improve its efficiency of ventilation and heat transfer, numerical simulation method is widely used [8]. In the 1970s, P.V.Nielsen[9] used CFD (Computational Fluid Dynamics) with the two-equation model to predict the indoor air flow. Then CFD was developed by Chen and Xu [10] into three-dimensional indoor flow field calculation successfully. Although the CFD technology has developed rapidly in recent years, and has shown very good performance in the calculation and prediction of flow field [11-13], but due to its complicated mesh generation processing, large differential equations to solve and other aspects, the CFD modelling of such big object like an indoor substation room becomes a great challenge [14, 15]. Since that, it is necessary to find a more efficient and more convenient method to solve this kind of CFD modelling of large-sized objects.
In this paper, we present a fast CFD method [16-18] and an automatic mesh generation system to predict the flow and heat transfer characteristics of the indoor substation room. And this new method has been used in the CFD modelling and optimization of a big indoor substation room.

2. Governing equations and algorithms

2.1. The flow model

(1) Boussinesq assumption

In an indoor substation room, the flow velocity is low and the temperature difference is small, the Boussinesq assumption can be used [19]. So in the model, the density and other physical properties of fluid parameters are taken as constants, and the impact of density change is put in the gravity item of momentum equation and energy equation. The continuity equation, momentum equation and energy equation are as follows:

\[
\frac{\partial u_i}{\partial x_i} = 0
\]

\[
\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho_0} \frac{\partial p}{\partial x_i} + \frac{\mu}{\rho_0} \frac{\partial^2 u_i}{\partial x_j^2} - g_i \beta (T - T_0)
\]

\[
\frac{\partial T}{\partial t} + u_j \frac{\partial T}{\partial x_j} = \alpha \frac{\partial^2 T}{\partial x_j^2}
\]

where, \( u \) is the speed, \( p \) pressure, \( T \) temperature, \( u \) air flow rate, \( \rho \) the air density, \( \mu \) the air kinematic viscosity. The subscript \( i \) which can be 1, 2, and 3 stands for the coordinate axis of \( x \), \( y \) and \( x \), and the positive direction of \( z \)-axis is defined as vertical upward, since that the three components of gravitational acceleration are \((0,0,-g)\). Based on the Boussinesq assumption, according to the reference temperature \( T_0 \) (the average temperature of the inlet and outlet air) and the reference pressure \( p_0 \) (the atmospheric pressure), the reference density \( \rho_0 \) can be calculated as following:

\[
\rho_0 = \frac{p_0}{RT_0}
\]

The physical parameters are assumed to be constants, and their specific values are related to the values of the reference temperature \( T_0 \). In the Equation 2, \( p \) is the relative pressure, which is the difference between the absolute pressure of the air and the reference pressure \( p_0 \), \( \beta \) is the coefficient of thermal expansion of the air. In equation 3, the definition of \( \alpha \) is as follows:

\[
\alpha = \frac{k}{\rho C_p}
\]

The density \( \rho \) can be calculated as:

\[
\rho = \rho_0 \left[ 1 - \beta (T - T_0) \right]
\]

Normally, the air flow in an indoor substation room is fully developed turbulence flow. There are three turbulence numerical schemes: Reynolds average, direct numerical simulation and large eddy simulation. Since it is not necessary to pay attention to all the flow states in the room, the faster scheme Reynolds averaging method is selected. To enclose the model equations, the Boussinesq eddy viscosity assumption (eddy viscosity model) is introduced, where the momentum equation and the energy equation become as follows:

\[
\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho_0} \frac{\partial p}{\partial x_i} + \left( \frac{\mu}{\rho_0} + \nu \right) \frac{\partial^2 u_i}{\partial x_j^2} - g_i \beta (T - T_0)
\]
\[ \frac{\partial T}{\partial t} + u_j \frac{\partial T}{\partial x_j} = \left( \alpha + \frac{\nu_T}{Pr_T} \right) \frac{\partial^2 T}{\partial x_j^2} \]  

(8)

where \( \nu_T \) is the turbulence viscosity, \( Pr_T \) is the turbulent Prandtl number.

Obviously, equations (7) and (8) have similar formalities, which can be written as a unified equation as follows:

\[ \frac{\partial \phi}{\partial t} + u_j \frac{\partial \phi}{\partial x_j} = \Gamma \frac{\partial^2 \phi}{\partial x_j^2} + S + G \]  

(9)

where \( S \) is the source term and \( G \) is the pressure term.

(2) Turbulence flow model

For the turbulence model, Chen and Xu [10] proposed a zero-equation model with high calculation performance in 1998:

\[ \nu_T = 0.03874VL \]  

(10)

where, \( V = \sqrt{u_1^2 + u_2^2 + u_3^2} \), length scale \( L \) is the shortest distance from the wall.

2.2. Governing equation solving algorithm

The finite volume discrete based on the grid center is used for the governing equations. All the flow field variables, such as velocity and pressure, stand for the values at the center of the grid [15]. Integrate Equation1, Equation7 and Equation8, and using the Gauss formula, we can get:

\[ \int_{cs} u \cdot dS = 0 \]  

(11)

\[ \frac{\partial}{\partial t} \int_{cs} udV + \int_{cs} uu \cdot dS = -\frac{1}{\rho_0} \int_{cs} pdS + \int_{cs} \nu \nabla u \cdot dS + \int_{cs} [-g \beta (T - T_0)] dV \]  

(12)

\[ \frac{\partial}{\partial t} \int_{cs} TdV + \int_{cs} Tu \cdot dS = \int_{cs} \alpha \nabla T \cdot dS \]  

(13)

For the continuous equation and the momentum equations, the value at \( n+1 \) time step can be gotten from the time step \( n \) using the stepwise projection algorithm, that is, from the time step \( n \) to the convection-diffusion intermediate step, the momentum equation with no pressure gradient term is solved. The second-order Adams-Bashforth scheme is used for the convective terms, and the implicit Crank-Nicolson scheme is used to the viscosity term. The semi-discrete scheme for convection-diffusion equations on each cell can be written as:

\[ \int_{cv} \frac{u^* - u^n}{\Delta t} dV = -\frac{1}{2} \left( \int_{cs} 3u^n V^n \cdot dS - \int_{cs} u^{n-1} V^{n-1} \cdot dS \right) + \frac{1}{2} \left( \int_{cs} \nu \nabla u^* \cdot dS + \int_{cs} \nu \nabla u^n \cdot dS \right) + \int_{cv} [-g \beta (T^n - T_0)] dV \]  

(14)

where \( V \) is the velocity vector at the center of the cell, \( V^* \) is calculated by the interpolation of \( u^* \) at the center of the cell. The pressure correction can be calculated as follows:

\[ \int_{cv} \frac{u^{n+1} - u^*}{\Delta t} dV = -\frac{1}{\rho_0} \int_{cv} \nabla p^{n+1} dV \]  

(15)

The final velocity field must satisfy the continuous form of the integral form, ie:

\[ \int_{cs} V^{n+1} \cdot dS = 0 \]  

(16)

And the relation between the pressure and velocity can be defined as:

\[ \int_{cs} \nabla p^{n+1} \cdot dS = \frac{\rho_0}{\Delta t} \int_{cs} V^* \cdot dS \]  

(17)
This is actually the integral form of the pressure Poisson equation. Then, after the intermediate step is completed, the pressure at \( n+1 \) step can be obtained by solving the above equation, then \( u^{n+1} \) and \( V^{n+1} \) are updated by the pressure correction step:

\[
\begin{align*}
  u^{n+1} &= u^* - \frac{\Delta t}{\rho_0} \left( \nabla p^{n+1} \right)_{cc} \\
  V^{n+1} &= V^* - \frac{\Delta t}{\rho_0} \left( \nabla p^{n+1} \right)_{fc}
\end{align*}
\]

(18)

(19)

Here, the solution of the continuity equation and the momentum equation are completed, and the energy equation is solved similarly. Since there is no pressure gradient term, only the convection-diffusion intermediate step is needed.

2.3. Spatial discretization of the calculation domain

Normally an indoor substation room is a regular cuboid, so be structured grid is a good choice. How to solve the problem of the "obstacle" in the indoor substation room like the fuel tank, heat sink and oil pillow of the transformer is one of the key points. As shown in Figure 1, we discrete the whole space including the internal "obstacles" using structured grids first, then identify the "obstacles" and define an appropriate boundary conditions for them using the approach of Immersed Boundary (IB) method [20, 21].

![Figure 1. Discretization solution of the calculation domain](image)

To satisfy the above assumptions, the input data to be provided for the grid generation include:
1) Lengths of the indoor substation room along the x, y and z direction;
2) The number of internal obstructions, the coordinates of each obstacle;
3) The number of air inlets and outlets, index of the surface where each air inlet and outlet is located, the coordinates of the air inlets and outlets;
4) The numbers of grid cells in x, y and z directions separately.

2.4. The treatment of fins

In order to enlarge the heat transfer area, the fins are widely used in the transformer. However, due to its thin thickness, large number and complex structure, it is difficult to get its detailed information of flow and heat transfer. A simplified scheme of "fin group" is used to deal with the fin problem. The fins are grouped by their temperature regions.

In each region, two "large fins" and three flow passages are substituted for all fins and the flows in this area. The fins' grouping solution is shown in Figure 2. The division of the temperature five regions: Group 1: the low temperature area at the sides; Group 2: the high temperature region in the middle area; Group 3: the middle temperature region; Group 4: the low temperature regions in the middle area which close to the fan; Group 5: the high temperature area in the middle region.
3. Simulation results and analysis

3.1. The Object introduction and boundary definition
In order to verify the correctness and applicability of the model, we calculated an indoor substation with a transformer (63 MVA load and 277 kW heat release). The basic structure is shown in Figure 3, the length, width and height are 11.0m, 10.5m and 8.7m separately. There are three inlets and two outlets in the indoor substation room, and there are five fans under the radiators. Two exhaust fans are installed at the two outlets separately.

Figure 2. The fins’ grouping solution

Figure 3. Schematic of the main transformer chamber
The total number of grids is 92000, and the air temperature is 36 °C. The heat sink is set as a solid wall boundary condition (wall), the total heat area is 95.34 m², and the heat source is 277kW. The default settings for the rest of the surfaces are solid wall boundary conditions.

3.2. FFD Computational accuracy analysis

1) The experimental process

In order to get the experimental data, we directly test the running indoor substation room with special measuring equipment. For this system, the most important parameter is the temperature distribution, since that, a temperature measuring pillar is used. Structure diagram of the temperature measuring pillar is shown in Figure 4, and there are seven points to put thermocouples. The temperature measuring positions are shown in Figure 5, and we put the measuring pillar at these nine positions separately to get the temperature distribution. At the same time, the temperatures and air flow velocities at all the inlets and outlets are measured at the same time. And we use an infrared thermometer to measure the temperature of the solid walls.

![Figure 4. Temperature measuring pillar](image1)

![Figure 5. Temperature measuring positions](image2)

2) The results and analysis

The comparisons between the experimental and numerical data with the FFD model are shown in Figure 6. The data in Figure 6 are the temperatures close to the inlet 2, which is the no.2 site as shown in Figure 5. It can be seen from Figure 6 that the maximum error is about 2% compared to the experimental values.
3.3. Comparison and analysis of different schemes

We calculated the flow and heat transfer characteristics of the indoor substation room with two different schemes, and the results are shown in Figure 7. It’s clear from these two figures that there is more fresh air cooling the fins when the two below air intakes solution is used, that means using this solution can obtain higher efficiency of heat transfer.

![Streamline diagram of the two sides air intakes solution](image)

(a) Two sides air intakes solution

(b) Two below air intakes solution

The maximum temperature value and the average heat transfer coefficient are shown in Table 1. It also shows that the two below air intakes solution has higher heat transfer coefficient.

**Table 1. Comparison of heat transfer effects of two different air inlet modes**

| No. | Air supply way       | Maximum temperature, °C | Average heat transfer coefficient, w/(m²*K) |
|-----|----------------------|--------------------------|--------------------------------------------|
| 1   | Two lower air intakes| 47.41                    | 41.67                                      |
| 2   | Two side air intakes | 64.77                    | 23.55                                      |
4. Conclusions
A fast CFD model is used to analyze the flow and heat transfer characteristics of the indoor substation room and through the numerical and experiment analysis, it is found that:

1) The fast CFD simulation method proposed in this paper can effectively calculate the flow and heat transfer characteristics of the indoor substation room. And comparing with the measured data, it proves that the calculated results can meet the engineering precision requirements.

2) The rational arrangement of the air inlet can effectively improve the heat transfer performance of the whole system, and it is important for the safety of the transformer in the indoor substation room.

Because of the flexibility of parameter setting, this model can not only be applied to the analysis of ventilation and heat transfer of indoor substation rooms, but also to other similar objects.

Acknowledgments
This work is supported by Chongqing Electric Power Company of the State Grid Project (No.2016022).

References
[1] Zhang F, Huang X, Ding J 2016 Fluid Machinery 01 76 (in Chinese)
[2] Wang J, Zhang F, Xing H 2016 Machinery Design & Manufacture 03 45 (in Chinese)
[3] Yang H, Liu X, Xu L 2016 J. Computer Applications S1 188 (in Chinese)
[4] Zhao B, Li X, Yan Q 2000 HV&AC 05 33 (in Chinese)
[5] Wang H, Jia L 2009 Distribution & Utilization 06 46 (in Chinese)
[6] Zhou X, Duan Y 2013 Inner Mongolia Petrochemical Industry 07 73 (in Chinese)
[7] Gao X, Li W, Song H 2012 J. civil architectural & Environmental Engineering 34(S2) 117 (in Chinese)
[8] Mo W, Zeng W 2004 Guangdong Power Transmission Technol. 5 27 (in Chinese)
[9] Patankar S V, Spalding D. B 1983 Numerical Prediction of Flow, Heat Transfer, Turbulence and Combustion (Oxford: Pergamon Press) PP 54-73
[10] Chen G, Xu W 1998 Energy and building, 2 137
[11] Zhang S, Morita K, Shirakawa N 2009 Journal of Power & Energy Systems 3(3) 313
[12] Leifsson L, Koziel S 2015 Journal of Computational Science 10 45
[13] Sun Z, Chaichana T 2016 International Journal of Cardiology 210 28
[14] Boncinelli P, Rubechini F 2004 J. Turbomachinery-Transactions of the ASME 126(2) 268
[15] Ferziger J. H., Peric M 2002 Computational Methods for Fluid Dynamics 3rd Edition (Berlin: Springer)
[16] Jin M, Zuo W, Chen Q 2012 Numerical Heat Transfer Part B-Fundamentals 62(6) 419
[17] Jin M, Zuo W, Chen Q 2013 Numerical Heat Transfer Part A-Applications 64(4) 273
[18] Jin M, Chen Q 2015 International Journal of Numerical Methods for Heat & Fluid Flow 25(1) 2
[19] Song S, Guo X 2012 Chinese Quarterly of Mechanics 1 60 (in Chinese)
[20] Gorse Y, Iollo A, Telib H 2012 Journal of Computational Physics 231(23) 7780
[21] Ye T., Mittal R., Udaykumar H. S 1999 Journal of Computational Physics 156(2) 209