Development and demonstration of CO₂ flow numerical calibration system – part I

Zhipeng Xue *, Yonghao Liao and Junfeng Xu
Huadian Electric Power Research Institute, Hangzhou Zhejiang, China

*Corresponding author e-mail: xuezhipeng@126.com

Abstract. This project takes the flue at the tail of Fujian Kemen Power Generation Company as the research object. Due to the diversified forms of the tail flue, the installation position of the flow measurement equipment affects its flow calibration coefficient. Through the computational fluid dynamics method, a numerical simulation model is established to perform numerical calculation on the entire flue. Determine the calibration coefficients of flowmeters in different positions and in different forms to achieve more accurate measurement of flue flow. The project is verified in combination with the actual site situation, and the numerical model of ultrasonic flowmeter is used to verify the accuracy of the numerical model. Based on numerical simulation and on-site verification, a flow calibration system combined with simulation analysis is developed. The system implements the input flue model, and the measurement point location and measurement form are calculated to generate a flow calibration report. The report includes recommended measurement methods, recommended measurement points, and solutions. Calibration factor, precise flow value, etc[1-3].

The project will eventually form a demonstration system and the development and demonstration of a CO₂ flow numerical calibration system.

1. Introduction

This project takes the flue at the tail of Fujian Kemen Power Generation Company as the research object. Due to the diversified forms of the tail flue, the installation position of the flow measurement equipment affects its flow calibration coefficient. Through the computational fluid dynamics method, a numerical simulation model is established to perform numerical calculation on the entire flue. Determine the calibration coefficients of flowmeters in different positions and in different forms to achieve more accurate measurement of flue flow. The project is verified in combination with the actual site situation, and the numerical model of ultrasonic flowmeter is used to verify the accuracy of the numerical model. Based on numerical simulation and on-site verification, a flow calibration system combined with simulation analysis is developed. The system implements the input flue model, and the measurement point location and measurement form are calculated to generate a flow calibration report. The report includes recommended measurement methods, recommended measurement points, and solutions. Calibration factor, precise flow value, etc[1-3].

The project will eventually form a demonstration system and the development and demonstration of a CO₂ flow numerical calibration system.

2. Technical requirements

This project is based on the development of the carbon emission testing project of Fujian Kemen Power Generation Co., Ltd., and carries out the following work for the tail flue:

2.1. Demonstration simulation

Simultaneous calibration flow of simulated nozzles for special flue ducts to ensure the accuracy of the model. As a result, the flow calibration coefficient of the actual measurement point (according to the actual flow measurement method, calculated deviation from the known flow) is required, a variable load flow coefficient curve is generated, and the recommended flow measurement point scheme, recommended measurement method or rectification opinions under normal load are given. The simulation comparison results are given for different principles of pitot tube, matrix flowmeter,
ultrasonic flowmeter and other flow measurement methods. Analyze the influence of the bend pipe, measuring position, flue gas and other factors on the calibration results.

2.2. On-site verification
According to the field measurement data of Huadian Kemen Power Generation Co., Ltd., in particular, the comparison between the matrix flowmeter and the ultrasonic flowmeter (installed under non-standard installation conditions) under the condition of special structure flue. Through the actual measurement and simulation results to further modify the model accuracy.

2.3. System development
For the above design method and analysis process, for a single product, the design cycle is already very long. However, on the basis of fully grasping the design methods and application simulation software, it is possible to develop products quickly. According to the above research content, its software mainly includes the following aspects: Based on the demonstration simulation, the application system is developed. After inputting the size of the flue, the location of the measuring point, and the measurement method of the unit to be tested, it can automatically measure the flow rate of different power plants and different principles. Generate calibration coefficient curve, automatically generate recommended schemes for load measurement points, recommended schemes for measurement methods, etc. The design interface meets the above reporting requirements.

2.4. Output results
Based on the existing test data, collate and summarize for data analysis; complete the flow calculation process; provide the final Chinese report. Form system software delivery.

3. CFD method
The basic idea of computational fluid dynamics CFD is to replace the continuous field of physical quantities in the time and space domains with a series of finite discrete point variable value sets, and establish the field variables at the relevant discrete points through certain principles and methods. The relationship between the algebraic equations, and then solve the algebraic equations to obtain the approximate value of the field variables.

3.1. Governing equation
In this report, the flow field solution is obtained by solving the N-S equation. Considering that the fluid medium is liquid, it can be regarded as incompressible, and the heat exchange amount can be ignored, so the energy equation is not solved. Equation 1 is the continuity equation, and equations 2~4 are the dynamic equations.

\[
\frac{\partial \rho}{\partial t} + \text{div}(\rho \mathbf{V}) = 0 \tag{1}
\]

\[
\frac{\partial (\rho u)}{\partial t} + \text{div}(\rho u \mathbf{V}) = \text{div}(\mu \text{grad } u) - \frac{\partial p}{\partial x} + S_u \tag{2}
\]

\[
\frac{\partial (\rho v)}{\partial t} + \text{div}(\rho v \mathbf{V}) = \text{div}(\mu \text{grad } v) - \frac{\partial p}{\partial y} + S_v \tag{3}
\]

\[
\frac{\partial (\rho w)}{\partial t} + \text{div}(\rho w \mathbf{V}) = \text{div}(\mu \text{grad } w) - \frac{\partial p}{\partial y} + S_w \tag{4}
\]

In the formula, \( \rho \) is the density, \( u, v, w \) is the velocity component in three directions \( x, y, z \), \( \mu \) is the dynamic viscosity, \( p \) is the pressure acting on the microelement, and \( S_u, S_v, S_w \) is the source term.
3.2. Numerical Discrete

The calculation uses the finite volume method to solve the RANS equation. The finite volume method (FVM) is a widely used discrete method in the field of CFD; the basic idea is: divide the spatial calculation area into a grid, and make each grid point have a mutual The control volume is not repeated, and the partial differential equation to be solved is integrated for each control volume, thereby obtaining a set of discrete equations.

In this calculation report, the convection term dispersion adopts the second-order upwind style, and the time dispersion adopts an implicit two-time step to solve the transient solution. The calculation uses SIMPLE algorithm solver, VOF multiphase flow model.

The SIMPLE algorithm uses the mutual correction relationship between pressure and velocity to force the conservation of mass and obtain the pressure field. If the pressure field \( p^* \) is used to solve the momentum equation, the surface flow \( J^* \) obtained from the continuity equation is:

\[
J^*_f = J^*_f + d_f (p^*_{c0} - p^*_{c1})
\]

It does not satisfy the continuity equation. Therefore, the correction term \( J'_f \) is added to the surface flow velocity \( J^*_f \) to correct the mass flow velocity \( J_f \):

\[
J_f = J'_f + J^*_f
\]

At this time, the continuity equation is satisfied. SIMPLE assumes that \( J'_f \) is written as follows:

\[
J'_f = d_f (p^*_{c0} - p^*_{c1})
\]

Where \( p^* \) is the unit pressure correction.

The SIMPLE algorithm substitutes the flow correction equation into the discrete continuity equation to obtain the discrete equation for the pressure correction \( p^* \) in the unit.

\[
a_p p' = \sum_{nb} a_{nb} p'_{nb} + b
\]

Among them, the source term \( b \) is the net flow rate into the unit.

\[
b = \sum_{f} J^*_f
\]

The pressure correction equation can be solved by algebraic multigrid method. Once the solution is obtained, use the following equation to correct the cell pressure and surface flow velocity:

\[
p = p^* + a_p p'
\]

\[
J_f = J'_f + d_f (p^*_{c0} - p^*_{c1})
\]

Here, \( a_p \) is the pressure sub-slack factor (please refer to the introduction of sub-slack). The corrected surface velocity \( J_f \) satisfies the discrete continuity equation uniformly in each iteration.

1.3.3 Boundary conditions

The boundary conditions are generally divided into three categories:

The first type of boundary condition is the Dirichlet boundary condition or Dirichlet condition, which directly specifies the value on the boundary, and this value can change with time.

The second type of boundary condition is the Neumann boundary condition or Neumann condition. It does not directly specify the value on the boundary, but specifies the value of the gradient on the boundary. For three-dimensional problems, the gradient takes the derivative of the normal direction of the boundary.

The third type of boundary condition neither directly stipulates the value on the boundary nor the value of the normal derivative on the boundary, but specifies a linear relationship between them.

In this report, the boundary conditions used for CFD calculations are the object boundary conditions, the wall surface meets the no-slip boundary condition, and the velocity on the wall surface is zero, namely:

\[
u_{wall} = v_{wall} = w_{wall} = 0
\]
4. Three-dimensional model establishment
According to the tail flue model provided by Fujian Kemen Power Generation Company, a three-dimensional calculation model is established as follows. The model includes the overall structure of the model, the inlet deflector, the defogger deflector, and the demister part.

Figure 1. Overall 3D model.
Figure 2. Overall 3D perspective

Figure 2 is a perspective view of the flue, which contains three parts, the inlet guide plate, the defogger guide plate, and the demister.

Figure 3. Detail of deflector and defogger
Figure 4. Overall grid diagram

Figure 3 shows the internal details of the flue.

4.1. Calculating model to build grid
According to the establishment of the geometric model, the calculation grid is divided as shown in the figure below. In order to ensure the calculation accuracy, the calculation grid is constructed with sufficient density to establish a calculation model grid of 15 million. The overall grid model is as follows:

Figure 5 Grid detail
Figure 6. Sectional view of the grid

The above is the grid diagram of the calculation model. According to the maximum flow velocity in the flow field <20m/s, the calculation of the cross-sectional size is about 10m. Set the computing grid size 0.1m to encrypt in the local and boundary layers, and finally the total number of grids is 15 million.
5. Results of numerical analysis of flue

The flue gas flow rate of thermal power plants has always been mainly in the form of traditional point measurement and line measurement. The outstanding problems such as poor accuracy and weak representativeness of monitoring data have long existed. The advantages of the above matrix flowmeter in practical applications are obvious. The following part uses the numerical simulation method to calculate the flow of the flue structure at the tail of Kemen Power Plant, extract the internal velocity and pressure distribution to provide the monitoring results of the flowmeter.

5.1. Internal mobility

Extract the internal flow of the flue at a flow rate of 20%-100%, and the picture results show that the instability of the internal flow of the flue at a constant flow rate increases as the flow rate increases. That is, the low-flow small flue is more stable inside, while the high-flow flow fluctuates greatly. The following are the screenshot results under different traffic.

5.1.1. 20% liquidity

![Figure 7. Cross-sectional velocity distribution at 20%](Image)

5.1.2. 40% (1440t/h) flow

![Figure 8. Cross-sectional velocity distribution at 40%](Image)
5.1.3. 60% (2160 t/h) flow

![Cross-sectional velocity distribution at 60%](image1)

**Figure 9.** Cross-sectional velocity distribution at 60%

5.1.4. 80% (2880 t/h) flow

![Cross-sectional velocity distribution at 80%](image2)

**Figure 10.** Cross-sectional velocity distribution at 80%
5.1.5. 100% (3600t/h) flow

![Figure 11. Cross-sectional velocity distribution at 100%](image)

5.2. Analysis of single-point flowmeter
According to the internal situation of the flow field, a section is set at the tail of the flue to extract the detection points. The top view of the section is as follows:

![Figure 12. Overhead cross-sectional velocity](image)

Figure 12.: The cross-sectional velocity distribution diagram in the top view. It can be seen that the flow stabilization section is affected by the demister and the tail bend. Multiple detection points are arranged here for analysis. There are two ways
5.2.1. Single cross section, multiple points (9×9 groups)

Figure 13. Probe point arrangement 1

5.2.2. Arrange 5 groups of 4×4 groups of probe points at different positions

Figure 14. Probe point arrangement 2
5.2.3. *Analysis of detection method 1*. According to the detection method 1, the data of 81 points in the calculation process is extracted, and the result is as follows:

![Figure 15. Distribution of the results of probe mode 1](image)

In Figure 15, the abscissa is the detection point number, a total of 81 detection points. The ordinate is the pressure difference measured at each point. Because the pressure is fluctuating, the scatter results walk in a range. It can be seen from the figure that due to the positional relationship; the pressure value of each probe point differs greatly. The theoretical differential pressure value is 81.09Pa. The maximum differential pressure measured at the probe point is 142-155Pa. The pressure difference at the minimum probe point is 2-23Pa. Most probe points are in the range of 75-115Pa.

6. **Conclusion**

The following conclusions can be drawn based on the above cross-sectional velocity distribution:

1. At a steady flow rate, the flow rate still fluctuates slightly, and as the flow rate increases, this fluctuation situation intensifies.
2. Affected by the demister, the flue gas flows out of the demister and there is a wake or vortex phenomenon. In this area, the velocity distribution deviation is large. This result determines that the deviation of single-point flowmeter in this flue is unstable.
3. There must be sufficient distance behind the defogger for the arrangement of the flowmeter.
4. The above conclusions are determined through data detection points.
5. According to the above results, the placement of a single probe in this flue cannot guarantee accurate measurement of flue flow. The reason is that the flue structure has a great influence on the flow velocity at different locations. The location cannot be guaranteed to be at a median value everywhere. In addition, the single-point pressure difference is affected by flow fluctuations, and its value has a large oscillation range, and the positive and negative deviations can reach 10-15%.

**References**

[1] Peiqing Liu, Huishen Duan, Wanli Zhao. Numerical investigation of hot air recirculation of air-cooled condensers at a large power plant. Applied Thermal Engineering, 29 (2009) 1927-1934.

[2] X.F. Gao, C.W. Zhang, J.J. Wei, et al.. Numerical simulation of heat transfer performance of an air-cooled steam condenser in a thermal power plant [J]. Heat and Mass Transfer. 45 (2009) 1423-1433.

[3] L.J. Yang, X.Z. Du, Y.P. Yang. Influences of wind-break wall configurations upon flow and heat transfer characteristics of air-cooled condensers in a power plant. International Journal of Thermal Sciences, 50 (2011)2050-2061.