STUDY ON MODELING METHOD IN MODULARIZING HVAC ELEMENTS FOR CFD SOFTWARE

Ken Fukada¹, Koji Sakai², Hiroki Ono³
¹Research & Development Center, Shinryo Corporation, Tsukuba, Japan
²School of Science & Technology, Meiji University, Kawasaki, Japan
³Environmental Science Research Lab., Central Research Institute of Electric Power Industry, Abiko, Japan

ABSTRACT
In this paper, we compared measurement results with the CFD analysis results using several of modeling methods for a four-way cassette type outlet. As a result, we compiled possible phenomenon in the case of inappropriate modeling methods. For example, the influence of the Coanda effect was overestimated than measurement results when we placed the outlet to the height that was the same as a ceiling and set diagonal airflow condition. Finally we described a modeling method of CFD parts which is appropriate for business and the design phase even if we carry out CFD analysis using a coarse mesh.

INTRODUCTION
The CFD analysis used in the field of architecture is utilized in each phase, such as business proposals and the design of HVAC systems in Japan, in recent years. For example, CFD analysis is used to reach the consensus by evaluating and comparing the analysis results at the ceiling blow-off system and the under floor air-conditioning system in the phase of designing the air conditioning system in the office. Proceeding to the construction phase, CFD analysis is used to determine the placement and number of the diffuser by performing CFD analysis considering the airflow characteristics of each diffuser. By substituting a model experiment for CFD analysis, it is reduced cost and time it took for the experiment. In addition, opportunities for design engineers themselves to perform CFD analysis are increasing year by year, and the reasons are that a processing capability of a computer has been improved and that a good operability software for CFD analysis has been widespread. It is considered that because of these, CFD analysis is now often used as a design tool of design engineers, and not an original technique of experts in R&D. However, the spread of gaps in analysis accuracy is a concern since many design engineers utilize CFD analysis. The difference tends to appear when setting boundary conditions, such as anemostat type air diffuser or four-way cassette type outlet, etc., for which some expert knowledge is required. It is also important to prepare the necessary data by surveying the specifications of HVAC elements, however if design engineers with little experience of CFD analysis don’t know the necessary data, trial and error is required. From these points of views, an academic committee has been established in Japan which conduct research and development in modularizing HVAC elements into “CFD parts” to make it more usable for HVAC design engineers. CFD parts is a package of information, geometry, boundary and field data file for reproducing HVAC element in CFD. “Subcommittee of Integration of BIM and Modularization of HVAC Elements for CAE software”, which is a part of “the Society of Heating, Air-Conditioning and Sanitary Engineers of Japan”, is now making out a draft of this CFD parts format and a guideline for making and use of it. There are two types of CFD parts. One is what we call “Micro parts”, it is part of the purpose of detailed comparison with the experiments and so on. Another is what we call "Macro parts", it is part of the purpose of approximate grasp of indoor thermal environment distribution at the design phase.

The micro parts are developed using the methods shown effectiveness in previous studies such as the box method, the prescribed velocity method and the momentum method. The macro parts are developed as parts that is input only the manufacturer’s specifications data without the need for experimental results. In the course of the study of macro parts for a four-way cassette type outlet, it was found that a large difference may appear in the analysis result due to the difference in reproducible way even if we input the outlets by geometric data read from the specification of HVAC elements based. In this study, for creating the macro parts which can be used at the design phase, we propose the modeling method. Firstly, we measured the airflow and temperature distribution in the four-way cassette type diffuser during isothermal state and heating state. Then, by comparing the CFD analysis and experimental results, it is verified to be an effective modeling method for the design phase. In addition, CFD parts are with a view to collaboration with Building Information Modeling (BIM). The data to perform CFD analysis for the three-dimensional shape of HVAC elements created by BIM software prepared in XML (Extensible Markup Language) format, CFD parts are handled as intermediate files to collaboration CFD software and
BIM software. For example, geometric shape, outlet placement, and outlet size, flow rate, flow angle, cooling and heating capacity, etc. are described. Incidentally, geometric shape, outlet position, and outlet size are the data prepared by simplified to adapt to CFD analysis.

OUTLINE OF MEASUREMENT

We measured temperature and wind velocity in the room that is used as exhibition room in the office building. Measurement is carried out with the circulation mode and heating mode. We measured Temperature and wind velocity in a steady state (circulation mode: February 27, 2014, heating mode: February 20, 2014).

Fig.1 shows the appearance and measurement area of the measured room. Nine four-way cassette type air conditioners are installed in the room. Vertical temperature distribution was measured in the center of the room by T-type thermocouple and thermography equipment (TH9100pro; Nippon Avionics Co., Ltd.). The surface temperature of the ceiling and the floor was measured by the thermography. Wind velocity distribution of the cross section was measured by three-dimensional ultrasonic anemometer; to measure the total 225 points in the 15 × 15 at 100mm intervals (WA-590; Sonic Corporation).

These measurement points are shown in Fig.2. The air volume of the air conditioner is calculated from the wind velocity data of suction port of 16 points were measured using a hot-wire anemometer (Model 6531; Kanomax Japan Inc.). The measurement situation by thermography is shown in Fig.3. Blow-off wind velocity of the air conditioner, to measure the main flow direction of the wind velocity at 24 locations shown in Fig.4. The wind direction was measured by visualizing the tufts. The wind velocity data is measured by 0.5 second intervals, it was used the average value of 10 seconds. It should be noted that since the vertical temperature distribution was the steady state at 9:30–10:50 for the heating mode (Fig.5), cross-section wind velocity distribution was described to F.L. +1.7m–2.5m.

CFD ANALYSIS OVERVIEWS

CFD analysis were solved using commercial software scSTREAMver.11 (Software Cradle Co., Ltd.). The approach was finite volume method and,
the air was treated as an incompressible fluid using Boussinesq approximation. The standard K-epsilon model was used for turbulence model. SIMPLEC method was adopted as an analytic algorithm. We were using the QUICK second-order accuracy difference scheme of the advection term. Mesh was a Cartesian coordinate system using a staggered mesh. These setting was assumed that the design engineers to use in practice. Iterations was set to 3,000 cycles, the convergence criterion was confirmed that the relative error of U,V,W-velocity, temperature, pressure, turbulent kinetic energy and turbulent dissipation rate was $10^{-4}$ or less. Fig.6 shows the calculation domain, which only covers around the air conditioner in the center of the room, because it is the same operation also for the eight air conditioners around the air conditioner.

**Modeling method for a four-way cassette type outlet**

Fig.7-a) shows the modeling method of the outlet of the air conditioner. It described the results of the 7 cases in total in this report. We placed the outlet boundary to the height that was the same as a ceiling in Case1. In Case2, we placed the outlet boundary apart 0.05m in vertical distance from the ceiling height and, the thickness of the air conditioner were set to 0.05m. In Case3, we align the edges of the outer shape of the air conditioner and the outlet boundary. In Case4-1, 4-2, 5-1 and 5-2, we set the outlet boundary in the vertical and horizontal faces (Fig.7-b). In case5-1 and 5-2, the outlet condition of the vertical face was set apart the same size as the outlet boundary from the ceiling height. In Case4-2 and 5-2, it was conducted by the model in the same size of the vertical and horizontal outlet boundaries for sensitivity analysis.

---

**Fig.5 Measurement results for vertical temperature distribution in the center of room**

**Fig.6 Calculation domain**

**Fig.7-b Position of outlet boundary for case4-1 and case5-1**

**Width for vertical face** : $W_v = W \times \sin \theta \cos \theta$

**Width for horizontal face** : $W_h = W \times \sin \theta$

**Fig.7-a Modelling method for four-way cassette type outlet**: above diagram is perspective, below diagram is elevation
Boundary conditions
Table 1 shows boundary conditions. The CFD analysis was conducted using the measured airflow rate of air conditioner, wind direction, outlet air temperature and wall surface temperature. The boundary conditions are presented in Table 1. The airflow rate of each outlet is to distribute the airflow rate of the air conditioner based on the wind velocity value of the measured inlet. The vertical surface was set a symmetry surface boundary condition. We set the convective heat transfer coefficient as the amount of heat balance by the amount of heat and the structure load caused by air conditioner is taken, so we conducted steady-state analysis.

Dividing mesh
The calculation domain contained 1,098,580 cells in case1, 2 and 3. The domain contained 1,222,080 cells in case4-1, 4-2, 5-1 and 5-2. Fig.8 shows the mesh

<table>
<thead>
<tr>
<th>Measurement</th>
<th>Case1</th>
<th>Case2</th>
<th>Case3</th>
<th>Case4-2</th>
<th>Case5-2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fig.9 Measurement and CFD analysis results during circulation mode (Wind velocity and velocity vector distribution [m/s])</td>
<td>Fig.8 Computational grid for case1</td>
<td>a) Plan around the outlet</td>
<td>b) Elevation (Whole area)</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
view of the case1. The mesh of the outlet was divided into three parts (0.007m). The average mesh size in the domain was 0.1m, and the maximum mesh size in the domain was set to 0.25m.

RESULTS AND DISCUSSION

1) Circulation mode

Fig.9 shows the wind velocity distribution of the measurement and analysis results. Since the change from case4-1 to case4-2 (from case5-1 to case5-2 as well) did not have a significant influence on the analysis results, it described only the results of the case4-2 and 5-2. Airflow blown out diagonally was attracted to the suction port of the air conditioner in the measurement results. In case1 and 2, blown the airflow crawled the ceiling surface, so the results showed a different result from the measurement result. Since the Coanda effect of the ceiling surface is overestimated, it considered airflow becomes easier to crawl ceiling. The factors are probably due that it is difficult to reproduce the flow near the ceiling using k-ε turbulence model and the CFD analysis was conducted using structured mesh. In case3, 4 and 5, the analysis results showed the same tendency as the measurement results. Fig.10 shows the correlation of wind velocity (horizontal component: (x), vertical component: (y)).

Table 2 $R^2$ between measurement and CFD analysis results during circulation mode

<table>
<thead>
<tr>
<th>case</th>
<th>x-velocity</th>
<th>z-velocity</th>
<th>U-velocity</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>0.40</td>
<td>0.31</td>
<td>0.37</td>
</tr>
<tr>
<td>4-1/4-2</td>
<td>0.35/0.39</td>
<td>0.35/0.51</td>
<td>0.37/0.56</td>
</tr>
<tr>
<td>5-1/5-2</td>
<td>0.58/0.63</td>
<td>0.50/0.74</td>
<td>0.52/0.71</td>
</tr>
</tbody>
</table>

Table 3 $R^2$ between measurement and CFD analysis results during heating mode

<table>
<thead>
<tr>
<th>case</th>
<th>x-velocity</th>
<th>z-velocity</th>
<th>U-velocity</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>0.59</td>
<td>0.68</td>
<td>0.61</td>
</tr>
<tr>
<td>4-1/4-2</td>
<td>0.81/0.65</td>
<td>0.38/0.25</td>
<td>0.63/0.49</td>
</tr>
<tr>
<td>5-1/5-2</td>
<td>0.64/0.71</td>
<td>0.38/0.39</td>
<td>0.50/0.58</td>
</tr>
</tbody>
</table>

Fig.10 Correlation of wind velocity between measurement and CFD analysis results during circulation mode

Fig.11 Measurement and CFD analysis results during heating mode (Wind velocity and velocity vector distribution [m/s])
component: (z) between measurement and analysis results. Table 2 shows the measurement and the CFD of coefficient of determination (R²) in each case. The result in case 5 shows the correlation was high. R² in case 3 and 4 were lower than R² in case 5, it is due to the influence of the suction air flow of the air conditioner has been overestimated.

2) Heating mode

<table>
<thead>
<tr>
<th>Measurement by thermography</th>
<th>Case1</th>
<th>Case2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case3</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Case4</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Case5</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

![Fig.12 Measurement and CFD analysis results during heating mode](image)

![Fig.13 Correlation of wind velocity between measurement and CFD analysis results during heating mode](image)
ceiling. This is due to the fact that the outlet airflow was excessively influenced by the Coanda effect and crawled the ceiling. The wind velocity and temperature distribution of case3, 4 and 5 showed almost the same trend with the measurement results. Fig.13 shows the correlation of wind velocity between measurement and analysis results. Table 4 shows the measurement and the CFD of R² in each case. In case3 and 5 had higher R² in much the same way. However, the outlet airflow on CFD analysis is expected to have received greater than the actual influence of the buoyancy. This is because, the correlation coefficient of case3 was higher than in the circulation mode, and the R²(z) of case5 was lower than in the circulation mode. In other words, it considered that the R² of case3 was higher due to cancel the calculation errors related the attraction by the suction airflow, and the buoyancy by the outlet airflow of temperature.

From the above, it is useful to modeling method of case5-2, when the wind direction of the outlet is around 45°. However, it is additionally required verification for the wind direction of the outlet at which begins affected by the Coanda effect and to be short circuit to the suction port of the air conditioner. We need to understand the limits of the wind direction can be reproduced by the same modeling method.

CONCLUSION

In this paper, we measured the temperature and the wind velocity distribution in the office where the four-way cassette type outlet installed during isothermal and non-isothermal state. We created seven macro parts using the geometric data read from the specifications, and performed CFD analysis. By comparing the measurement results and the analysis results, the following findings were obtained.

- If we placed the outlet to the height that was the same as the ceiling, the Coanda effect is overestimated and the analysis results show different results from the actual airflow field.
- Better modelling method for macro parts was the method that was how to set up the end of the outlet the external dimensions, and to set separately the boundary conditions of the outlet to the same size vertical plane and horizontal plane. Analysis results of using this macro parts were generally consistent with the measurement results of the airflow distribution during isothermal and un-isothermal state.

ACKNOWLEDGEMENT

This study was received valuable advice to members of "Subcommittee of Integration of BIM and Modularization of HVAC Elements for CAE software". We express gratitude to the members.

REFERENCES


