1. Introduction

In recent years, due to high performance requirements of air conditioning systems, the complexity of HVAC internal flow path is progressing, i.e. leaf/right independent process control type system for automobile, and rear air conditioning for the rear seat passengers, etc., furthermore, despite speeding up in development, the demand for quality assurance, performance improvement and cost reduction is increasing. Therefore, in the development project of HVAC units, it is necessary to ensure target performance at an early stage and to make efforts to improve quality.

In order to achieve the demand target in performance improvement at an early stage of development, it is indispensable to grasp the information on flow and temperature field using CFD simulation. However, in the case where warm air and cool air mixed inside HVAC units, if utilization of a default simulation condition in the commercial CFD software to predict temperature at duct outlet, in which the default setting of Prandtl number is usually set to about 0.7 to 0.9 as laminar airflow, there is a big difference between the simulation result and the measured data, and the quantitative evaluation is very difficult and a hard task, because the airflow inside HVAC units has a very complex turbulence vortex structure, and moreover the structure of the turbulence airflow changes with the variation of the gap between the temperature control damper and the associated shut rib, due to the various air mixing opening degrees. Therefore, the determination of the turbulent Prandtl number is a key point for improvement of CFD simulation accuracy.

According to our previous studies, it has been confirmed that some simulation results for temperature are close to the measured valves by adjusting the turbulent Prandtl number to an arbitrary constant of 0.7 or less, and it is also known that the precision of simulation results deteriorates depending on the position of measurement point, the reason for this is that the turbulent Prandtl number is not a constant value but is considered to be change with variation of the flow and temperature field. For instance, Kitada et al. determined a function of the turbulent Prandtl number with eddy viscosity coefficient and the temperature gradient by an experimental studies of air mix basic model, and applied in CFD simulation to improve temperature prediction accuracy.

In the study, a functionalized Pr is proposed by theoretical analysis using the numerical results of velocity and the measured data of temperature in a
simple air mix model. In addition, the functionalized Prt was applied in CFD simulation to predict the temperature of the air flow inside HVAC units, and the simulation accuracy will be verified by the comparison between the experimental data and the simulated result.

2. Turbulent Prandtl Number

2.1. Definition of Turbulent Prandtl Number

The turbulent Prandtl number is a classical approach for the heat transfer problem in turbulence, similarly as in molecular Prandtl number, which is defined as the ratio between the momentum eddy diffusivity $\varepsilon_m$ and the heat transfer eddy diffusivity $\varepsilon_h$, which is

$$Pr_t = \frac{\varepsilon_m}{\varepsilon_h}$$

(1)

using Boussinesq relation, the shear stress $\tau$ and heat flux $q$ can be written as follows:

$$\frac{\tau}{\rho} = (v + \varepsilon_m) \frac{du}{dy}$$

(2)

$$\frac{q}{\rho C_p} = (\alpha + \varepsilon_h) \frac{dT}{dy}$$

(3)

There are basically two methods by which $Pr_t$ may be evaluated experimentally.

In one method, utilizing experimental time-average velocity and temperature profiles together with the integrated Reynolds equations. From equations (2) and (3) to obtain

$$Pr_t = \frac{\tau/\rho}{du/dy - 1}$$

(4)

$$Pr_t = \frac{q/\rho C_p}{dT/dy - 1}$$

thus, if the momentum and heat flux variation with the distance from the wall are known, $\varepsilon_m$, $\varepsilon_h$, and hence $Pr_t$ can be evaluated.

Alternatively, direct measurement of the Reynolds transport terms ($\overline{u'v'}$, $\overline{T'v'}$) and utilization of the definition of the eddy diffusivity, which gives

$$\frac{\tau}{\rho} = \overline{u'v'} = \varepsilon_m \frac{du}{dy}$$

(5)

$$\frac{q}{\rho} = \overline{T'v'} = \varepsilon_h \frac{dT}{dy}$$

(6)

then substituting the Eqs. (5) and (6) into Eq. (1) yields

$$Pr_t = \frac{\overline{u'v'} \; dT \; dy \; dU}{\overline{T'v'} \; dy \; dU} = \frac{\overline{u'v'} \; dT}{\overline{T'v'} \; dU}$$

(7)

thus, if the velocity and temperature fields, and the turbulent fluctuating terms are known, the $Pr_t$ can be evaluated.

The latter method will be used here.

3. Investigation Using an Air Mix Model

3.1. Experimental Verification

As shown in Fig. 1, create a simple experimental model to imitate the air mix condition inside the HVAC units, two heat exchangers are placed in model, one heat exchanger with warm water generates a warm air flow, and mixing with a cold air flow is performed by a guide near the confluence on the downstream, multiple sensor pins are arranged in lattice pattern in air mix region.

![Air mix model for Experiment](image-url)
The experiment was carried out on a test bench that can be controlled the flow rate of air and temperature of warm water, the temperature of warm water is fixed and the inlet air volume is set in 3 cases. A multi-channel wind velocity and wind temperature sensor system is fixed on multiple sensor pins to measure the velocity and temperature field of the air mix region.

3.2. Validation by CFD

Since the wind velocity sensor is not able to measure Reynolds stress $\overline{u'v'}$ of the air flow field, after clarifying how much the CFD velocity result correlates with the measured data, the CFD results will be attempted to use instead of the measured data.

3.2.1. Simulation Conditions

The 3D-CFD model completely reflects the experimental model with the multiple sensor pins (see Fig. 2). The heat exchanger is defined as a porous body, and the permeability coefficient is calculated from the experimental data of air flow resistance. Additionally, the amount of heat generation per unit volume is inputted as a quadratic function of air velocity in the heat exchanger, and the maximum heating temperature is limited so as to be equal to the warm water temperature at inlet of heat exchanger.

In this research, the CFD simulation is carried out by commercial CFD software, and the simulation conditions are listed in Table 1.

<table>
<thead>
<tr>
<th>Fluid model</th>
<th>Air mix model</th>
</tr>
</thead>
<tbody>
<tr>
<td>State</td>
<td>Unsteady</td>
</tr>
<tr>
<td>Mesh Structure</td>
<td>All Polyhedral</td>
</tr>
<tr>
<td>Turbulence model</td>
<td>DES (SST k-ω)</td>
</tr>
<tr>
<td>Difference Scheme</td>
<td>2nd up wind scheme</td>
</tr>
<tr>
<td>Inlet volume flux ([m^3/h])</td>
<td>200, 300, 400</td>
</tr>
</tbody>
</table>

3.2.2. A Comparison of Simulation and Experiment

Figure 3 present the flow field and temperature field of CFD results in case of 400m$^3$/h, it indicates that the mixing region has a non-uniform temperature field, and at each measurement point in grid pattern, there are temperature and flow fields with different gradients, respectively.

A comparisons of velocity at each measured point between simulation and experiment are shown in Fig. 4-5, they show that the CFD results of velocity give a close agreement with the experimental data, the coefficient of determination $R^2$ equals 0.89 and the standard error of the mean is

![Fig. 3 Velocity and temperature of CFD at 400m$^3$/h](image)

![Fig. 4 Correlation between experiment and CFD part1](image)
17.8%, in the study, the error rate level is acceptable for determination of $P_{rt}$.

Equation (8) is applied to CFD simulation of the simple air mix model, and compared to the case of $P_{rt} = 0.9$, maximum error decreased from 10.8°C to 8.0°C within the interval $\pm 2\sigma$ as shown in Fig. 8, and the probability of occurrence within the interval

$$P_{rt} = aRe^bPr^c$$

where, the coefficients $a$, $b$ and $c$ are constant coefficient, respectively, they can be determined using separation of variables $Re$ and $Pr$ by simply taking the natural logarithm to Eq. (8).

Equation (8) is applied to CFD simulation of the simple air mix model, and compared to the case of $P_{rt} = 0.9$, maximum error decreased from 10.8°C to 8.0°C within the interval $\pm 2\sigma$ as shown in Fig. 8, and the probability of occurrence within the interval
±5°C was upgraded 15%, which improves from 68.2% for Prt = 0.9 to 83.6% for functionalized Prt. Consequently, utilization of the functionalized Prt is effective for improving the temperature simulation accuracy of air mix model.

5. Applied on CFD Simulation of HVAC Units

The functionalized Prt was employed to CFD simulation of HVAC units (see Fig. 9) for verifying its simulation accuracy, and Table 2 shows the conditions of CFD simulation. Here, the dimensionless temperature Td is defined by dividing the air temperature T by the heater warm water temperature Th.

Figure 10 presents the characteristics of temperature varied with respect to damper open degree changes in the Bi-Level (B/L) and the Heat Differential (H/D) modes of HVAC units, where solid line indicates measured data and dash line indicates simulation results, respectively. It shows that the inclination of the thermal control characteristics at each blowing outlet are obtained, and the tendency is approximately in agreement with experimental data.

In regards to the probability distribution of errors, in comparison to a default case of constant Prt = 0.9, maximum error decreased from 11.2°C to 7.9°C within the interval ±2σ as shown in Fig. 11, and the probability of occurrence within the interval ±5°C is 67.4% for Prt = 0.9 and 80.4% for functionalized Prt, which was upgraded 13.4%, this demonstrates

<table>
<thead>
<tr>
<th>Fluid model</th>
<th>Semi center HVAC</th>
</tr>
</thead>
<tbody>
<tr>
<td>State</td>
<td>Steady</td>
</tr>
<tr>
<td>Mesh Structure</td>
<td>All Polyhedral</td>
</tr>
<tr>
<td>Turbulence model</td>
<td>Realizable k-ε</td>
</tr>
<tr>
<td>Difference Scheme</td>
<td>2nd up wind scheme</td>
</tr>
</tbody>
</table>

Table 2 Simulation conditions

Fig. 9 HVAC model

Fig. 10 Thermal control characteristics

Fig. 11 Comparison of temperature
that the functionalized Prt has been effective for improving temperature prediction accuracy in CFD simulation of the HVAC units.

On the one hand, in the opening degree 70% of the H/D mode, although the simulation temperature (Td) of DEF was only 0.05 lower than the measured data and it showed a relatively good results, but on the other hand, in the opening degree 100%, the simulation temperature of DEF and R-HEAT were higher than the measured data, and that of HEAT showed a tendency to be lower conversely, it is considered that the correction of functionalized Prt is insufficient for variation of thermal transfer and diffusion, due to the reason that Re number of air toward the DEF becomes high and exceeds the range as shown in Fig. 12. This is probably due to the fact that the air mix model which used in the study, is relatively simple mixing with only one air mix guide, and the application range of functionalized Prt is the same length as the Prt distribution range (see Fig. 7).

In order to improve adaptability to different complex flow fields in each mode and opening degree of HVAC units, the coefficients a, b and c in Eq. (8) are considered to vary at a critical value of Re number, and further verification is necessary.

<table>
<thead>
<tr>
<th>A/M Position</th>
<th>Td</th>
<th>Re</th>
<th>Prt</th>
</tr>
</thead>
<tbody>
<tr>
<td>70%</td>
<td></td>
<td>High</td>
<td>1</td>
</tr>
<tr>
<td>100%</td>
<td></td>
<td>Low</td>
<td>0</td>
</tr>
</tbody>
</table>

Fig. 12 Flow field of HVAC unit in H/D mode

6. Conclusion

In the study, a functionalized Prt is obtained from a combination of experimental data for temperature and numerical results for velocity, and the turbulent Prandtl number (Prt) can be expressed as a function of both Reynolds number (Re) and molecular Prandtl number (Pr).

In addition, the functionalized Prt was employed to predict the temperature of the air flow at the outlet with CFD simulation of HVAC units. As a result, simulation results of the temperature field were in relatively good agreement with experimental data. This demonstrates that the functionalized Prt has been effective for improving temperature prediction accuracy in CFD simulation of the HVAC units.

7. Future Tasks

As mentioned in chapter 5, since only one simple flow field is discussed, and the applicability range to CFD simulation of HVAC units was limited, then in the future tasks, the applicability can be improved with increasing of discussion cases, such as the number of guides and variation of the temperature difference between air and warm water.

Moreover, it just set the wall heat transfer coefficient as a constant in CFD simulation in the study, however, the certainty of the constant has not been clarified. Therefore, the other future challenge is to elucidate the physical phenomenon concerning heat exchange between the external environment and HVAC case or the duct, and to identify the wall heat transfer coefficient for further improvement in simulation accuracy.

References

(2) Jun Onodera, etc.: Numerical Simulation of Flow-Induced Noise in Automotive HVAC Systems,
In the paper, a theoretical-based approach to identifying turbulent Prandtl number (Prₜ) in air mixing is challenged, and a function of non-unity Prₜ number is proposed. In addition, we are also planning to improve the function of Prₜ with increment of experimental conditions in the next step, and to expand its application range on CFD simulation of HVAC units. Finally, I would like to thank everyone for their guidance and cooperation during the research. (ITO)