Srebric, J. and Chen, Q. 2002. "An example of verification, validation, and reporting of indoor environment CFD analyses," *ASHRAE Transactions*, 108(2), 185-194.

An example of verification, validation, and reporting of indoor environment CFD analyses (RP-1133)

Jelena Srebric, Ph.D.
Member ASHRAE

Qingyan (Yan) Chen, Ph.D.*
Member ASHRAE

ABSTRACT

This paper illustrates a step-by-step process on how to use "The Manual for Verification, Validation, and Reporting of Indoor Environment CFD analyses" developed by ASHRAE. The details of the manual are further explained through an example of indoor environment modeling (an office with mechanical displacement ventilation). The verification, validation, and reporting do not have to be documented in a uniform format, but do have a very similar procedure. The emphasis of the verification and validation is problem dependent. The computational fluid dynamics (CFD) user plays an important role in the verification and validation. An accurate and successful simulation is a combined effort of the CFD code and the CFD user.

Keywords: Air flow, Measurement, Research report, Simulation, Space environment, Turbulence

INTRODUCTION

Computational fluid dynamics (CFD) technique has become a tool for indoor environment analyses since the 1970s, due to the development in computer programming and turbulence models. One of the pioneer CFD studies is from Nielsen (1974). CFD solves fluid flow, heat transfer, and chemical species transport. The parameters solved, such as air velocity, air temperature, contaminant concentrations, relative humidity, and turbulence quantities, are crucial for designing a comfortable indoor environment. This is because the design of appropriate ventilation systems and the development of control strategies require detailed knowledge of airflow, contaminant dispersion and temperature distribution in a building. In the past thirty years, the CFD technique has been applied with considerable success in indoor environment design and analyses as evidenced by numerous papers published in ROOMVENT (2000).

In order to help building engineers correctly and effectively perform indoor environmental modeling using CFD, some researchers have setup fine examples how to verify, validate, and report CFD results, such as Baker and Kelso (1990), Baker and Gordon (1997), Kirkpatrick (1998), Muramaki et al. (1997, 1998), and Nielsen (1995). However, those examples were for specific applications and may not be used as a general guide to verify, validate, and report CFD analyses of indoor environment. Chen and Srebric (2002) have recently developed a manual defining the steps necessary to verify, validate, and report CFD analyses in indoor environment applications through ASHRAE RP-1133. A more completed report is also available from ASHRAE (Chen and Srebric 2001).

The manual defined verification, validation, and reporting of results as follows:

Jelena Srebric is an Assistant Professor, Department of Architectural Engineering, The Pennsylvania State University, University Park, PA and Qingyan (Yan) Chen was a Visiting Professor at Welsh School of Architecture, Cardiff University, Wales, UK. Chen's current address: Room 5-418, 77 Mass. Ave., Cambridge, MA 02139. Phone: (617) 253-7714, Fax: (617) 253-6152, Email: qchen@mit.edu

- The *verification* identifies the relevant physical phenomena for the indoor environmental analyses and provides a set of instructions on how to assess whether a particular CFD code has the capability to account for those physical phenomena.
- The *validation* provides a set of instructions on how one can demonstrate the coupled ability of a user and a CFD code to accurately conduct representative indoor environmental simulations with which there are experimental data available.
- The *reporting of results* provides a set of instructions on how to summarize the results from a CFD simulation in such a way that others who see the results can make informed assessments of the value and quality of the CFD work.

The manual suggests that the decision to use CFD must be firmly based on realistic expectations of its performance, cost, and effort required. It is necessary to provide instructive materials on how to verify, validate, and report indoor environmental CFD analyses. The paper (Chen and Srebric 2002) or the report (Chen and Srebric 2001) recommends verifying and validating a CFD code for indoor environment modeling based on the following aspects: basic flow and heat transfer features, turbulence models, auxiliary heat transfer and flow models, numerical methods, assessing CFD predictions, and drawing conclusions. Although the format for reporting of CFD analysis does not necessarily have to be the same, it is necessary to include all the aspects used in verification and validation for technical readers. It can be simpler for non-technical readers.

This paper attempts to illustrate, step-by-step, the manual for a representative indoor environmental modeling application (an office with mechanical displacement ventilation). With the application, a building engineer can follow similar steps to perform correctly and effectively simulations of indoor environment by CFD.

CASE SETUP

This example concerns the design of a displacement ventilation system in an office as shown in Figure 1 to achieve an acceptable level of thermal comfort and indoor air quality. The thermal comfort is considered to be related to air velocity, air temperature, relative humidity, mean radiant temperature, turbulence intensity, clothing level, and activity level, etc. (ISO 1990, Fanger et al. 1989). Since the space studied is an office, the clothing level is assumed to be 1.0 clo for winter and 0.8 clo for summer. The activity level is 1.0 met. The criteria used for the evaluation of indoor air quality in an office can be airborne contaminant concentrations and particulate concentrations at different size. Considering the office is generic and has limited particulate sources, this study uses carbon dioxide (CO_2) distributions for evaluating indoor air quality. The designer therefore needs to determine the distributions of

* Airflow
* Temperature
* Relative humidity (water vapor)
* Environmental temperature
* Turbulence intensity
* Carbon dioxide concentration

VERIFICATION

This example involves turbulent airflow, heat transfer (conduction, convection, and radiation), and mass transfer (water vapor and CO_2 concentrations). Since a commercial CFD code is used, the first step is to verify whether the CFD code is capable of predicting the flow, heat transfer, and mass transfer in the proposed study. The verification is performed for the following aspects:

* Basic flow features
* Turbulence models
* Auxiliary heat transfer and flow models
* Numerical methods
* Assessing the CFD applications

Basic flow features

The flow in the office is affected by the heat transfer from building enclosures, the occupants, and other heated objects, as well as the air supply from diffusers. Therefore, the flow in the office is mixed convection associated with conductive, convective, and radiative heat and mass transfer. The parameters to be solved are air velocity, temperature, relative humidity, environmental temperature, and CO_2 concentration. From the CFD fundamentals, we know that the CFD code can solve the problem. This paper only illustrates the verification process for mixed convection instead of the other heat and mass transfer processes involved. .

Blay (1992) conducted experiments on mixed convection in a two-dimensional cavity as shown in Figure 2. Although the configuration is a laboratory model rather than an actual room, the flow is mixed convection that represents flow features found in the office. Therefore, Blay's case is used for verification of the CFD code to predict mixed convection.

The experiment maintained a temperature, T_w=15°C (59^{o}F), at the two vertical walls and the top wall while the floor was heated to a higher temperature, T_{fl} = 35.5°C (95.9^{o}F). An air jet with a temperature of 15°C (59^{o}F) was discharged horizontally into the cavity at a velocity that varied from 0.25m/s (50 fpm) to 0.57m/s (114 fpm). The Re number is 658 based on the inlet height or 38,000 based on the cavity height. The Ra number is $1.8x10^6$.

Turbulence models

Since the office is ventilated by mechanical ventilation and the study is for ventilation design, two extreme scenarios need to be considered: summer design condition and winter design condition. Therefore, a steady-state flow simulation is sufficient, because the steady state simulated a continuous hottest or coldest condition. For such a problem, we can choose Reynolds stress models or eddy viscosity models. The manual suggests starting a simple and popular model, such as the standard k-ε model. However, this turbulence model is general, but not universal, and it was not developed especially for indoor environment modeling. Therefore, this study selects the two-layer turbulence model from Xu and Chen (2000) that is especially suitable for indoor airflow simulation. In order to make such a decision, it usually takes several steps. In our case, the investigation first used the standard k-ε model and found the heat transfer calculated was not accurate. Then, low-Reynolds-number k-ε models were tested but the computing time was too long. The two-layer model could accurately predict heat transfer and uses nearly the same computing time as that of the standard k-ε model.

Although the case is simple, the CFD analysis must use some approximations. For example, the CFD analysis assumes non-slip condition for the velocity, constant temperature condition, and zero kinetic energy at walls. The dissipation at the inlet, ε_{in}, is set to zero. Those assumptions are based on the user's best guess. The user should not be afraid of making assumptions. Good assumptions can simplify the complex physical phenomena in the real world with negligible effect on the accuracy of the CFD prediction.

Auxiliary heat transfer and flow models

The mechanical ventilation in an office does not require the simulation of other heat transfer and flow processes. For example, the experimental data give the wall surface temperatures that have already consider radiation. A radiation model is not needed for this case. Therefore, no auxiliary model is used in this study.

However, the CFD simulation assumed a 6% of turbulence intensity and zero turbulence dissipation at the inlet, since no measured information is available. This assumption could lead to some errors in the simulation. The outlet was simulated with zero-pressure and zero-gradients for other variables solved. In this case, the first grid was in the laminar sub-layer so the temperature at the grid was fixed to be the same as the wall temperature.

Numerical methods

The office geometry and the majority of the objects in the office are rectangular. Although some objects are not rectangular, such as the occupants, it is not necessary to represent the human bodies by either body-fitted coordinates or an unstructured grid system, because the contribution of the geometrical form of the bodies on airflow is small. Therefore, a Cartesian coordinate system with structured mesh system is selected. With the Cartesian coordinates and structured mesh system, the numerical procedures with finite-volume method usually converge much faster.

The systematical refinement of the grid resolution is conducted in this study for the grid number from 35×32, 40×40, to 50×50. The grid distributions are not uniform (denser grid distribution near the solid surfaces to account for the large gradients of the variables and coarser grid in the center of the cavity). Figure 3 compares the measured mean velocity at x/L=0.5 with the computational results. The comparison indicates that the calculation with 35×32 grids already produces accurate results with a less than 3% difference, compared with finer grids; increasing the resolution does not yield a significant difference and improvement.

This flow is a steady state so that the time step is not an issue.

The upwind and hybrid differencing scheme and SIMPLE, SIMPLER, and SIMPLEST algorithms (Patankar 1980) are all tested in this application. The results and computing time with different schemes and algorithms are quite close for the two-dimensional verification model. However, the upwind scheme (which is the first-order differencing scheme) and SIMPLE algorithms seem to be more stable. Therefore, they are selected for the office design simulation.

The values selected for relaxation factors depend very much on user's experience. The values of relaxation factors (false time steps in this application), as shown in Table 1, can be different for different parameters. Since the complete system of the office model in this case is more complex than the two-dimensional mixed convection case, the relaxation factors may be quite different from those listed in the table.

It is also found in the verification process that if monitoring points are used to assess the convergence, they should be placed in a region with higher velocity instead at the core for this case, where the velocity is low. This is because the velocity at the core still changes over time even though the residual error gets very small. For such a simple case, the computation can be carried on until the residuals are much smaller than those suggested in the manual, for which the additional computing time required is insignificant.

Note that the discussion on numerical scheme, iteration, and convergence is very brief here despite the fact that it is one of the most difficult aspects of simulations. This is because the manual is not meant to provide instructions on how to obtain converged results. There are some general rules that apply and are detailed in many CFD textbooks. Very often, a user needs experience to obtain a converged solution.

Assessing the CFD predictions

In this two-dimensional verification case, the first step is to compare the general airflow pattern as shown in Figure 4. If a CFD code cannot correctly predict the airflow pattern, it is meaningless to compare the velocity, air temperature, humidity, and CO_2 concentration distributions. The figure shows that the CFD computation captured the major characteristics of the flow pattern observed in the experiment, such as the main eddy in the center of the cavity and the small eddy close to the inlet jet.

With the correct prediction of the airflow pattern, Figure 5 further compares the mean temperature and turbulent kinetic energy between the computed results and the experimental data (Blay et al. 1992) at x/H=0.5. The comparison in Figure 5 shows that (1) there is less than 1°C difference in temperature prediction and (2) the agreement on the predicted turbulent energy is not as good as the mean quantities, such as mean air velocity and temperature.

The verification case accounts for the key features and the simplest physical phenomena in the indoor space. Two problems of the CFD code are revealed: (1) The heat transfer from the floor is slightly under-predicted so that the computed temperature is slightly low, and (2) the two-layer model cannot accurately predict the turbulence. The errors can be larger when the CFD simulation is applied for a real indoor environment. In this particular case, the user accepts the errors and considers the results predicted by the CFD code to be satisfactory, because turbulence is a secondary comfort parameter compared with the air temperature and velocity. Therefore, the CFD code has the capacity of indoor environment modeling.

This section shows only one case of verification. It verifies only mixed convection problem. In practice, there are other flow features in an indoor space, such as jet flow and flow generated by purely buoyancy force. Those flow features, if important, should be verified accordingly.

VALIDATION AND REPORTING OF RESULTS

This paper combines the complete system validation with the reporting process, although the reporting of the CFD results should generally include both verification and validation. This section uses a complete indoor environment system as an example to demonstrate the procedure regarding how to use the manual to report CFD results.

Experimental description

The first part is a detailed description of the experimental setup with which other people can repeat the CFD simulation. The experimental data from Yuan et al. (1999) regarding the displacement ventilation in an office, as shown in Figure 6, is used. There is one supply diffuser, one exhaust, two occupants, two computers, two tables, two boxes, and six lamps in the room as modeled in Figure 6. The sizes, locations, and heat released of these items are listed in Table 2.

Since water vapor (relative humidity) and CO_2 transport has the same features, this study used a tracer gas, SF_6, to simulate all the mass transfer. The SF_6 simulated CO_2 from the two occupants as a rate of 40 ml/h on the top of the occupant, delivered through a porous sphere of 0.10 m. Note that this sphere is smaller than the CFD cell used, although the total source from the entire cell is the same (40 ml/h). The additional energy associated with the trace-gas was small and not considered in the simulation. The supply air temperature was 17^{o}C (62.6 ^{o}F) with zero SF_6 concentration, and airflow rate 4.0 ach that corresponded to a face velocity 0.09 m/s (18 fpm). The exhausted air temperature was 26.7^{o}C (80.1^{o}F) with SF_6 concentration of 0.42 ppm. The measured data also included wall surface temperatures. With such detailed information, a CFD model can be created. This case presents the same flow, heat transfer, and mass transfer features as the office to be designed (Figure 1). Therefore, it is a good choice.

The measured data that can be used for validation are the distributions of:

* Air temperature
* Tracer-gas concentration
* Mean velocity
* Fluctuating velocity of turbulence

These data will be presented in graphical charts for comparison with CFD results that will be reported later.

In the CFD report, it is important to present the errors in the measurements, such as:

* Velocity: The repeatability is 0.01 m/s (2 fpm) or 2% of the readings, and the anemometers used cannot reliably measure velocity when the magnitude is lower than 20 fpm (0.10 m/s).
* Velocity fluctuation: Error is unknown.
* Air temperature: ±0.04 K (0.8 ^{o}F), including the errors introduced by the data acquisition systems.
* Tracer gas concentration: ±10%

The error information is useful in assessing the quality of the experimental data. In addition, any uncertainties in the experiment should be stated.

Then, it is possible to report the uncertainties in the CFD simulation. For this particular case, the uncertainties are: air velocity 0.04 m/s, temperature 2 K (3.5 ^{o}F), and tracer gas concentration 0.25 ppm.

Turbulence model and auxiliary heat transfer and flow models

The second step is to describe the turbulence model used, which was the same two-layer model (Xu and Chen 2000) as in the model verification. Since it is not a well-known model, a description of the model is necessary.

The two-layer model consists of two turbulence models, a single k-equation turbulence model for near wall flow and the standard k-ε model (Launder and Spalding 1974) for the flow in outer-wall region. The criterion to switch the model from one equation to the other is based on y* value, the turbulent Reynolds number defined as

$$y^* = y_n \sqrt{k} / \nu \qquad (1)$$

where y_n = normal distance to the nearest wall

k = turbulent kinetic energy

ν = kinetic viscosity

If y^*<80 the single-equation model applies; otherwise, the standard k-ε model will be used. In the near-wall region, where y^*<80, the new one-equation model is used; i.e., the k is solved by

$$\frac{\partial k}{\partial t} + U_i \frac{\partial k}{\partial x_i} = d_k + P_k + G_k - \varepsilon \qquad (2)$$

where U_i = mean velocity in x_i direction

d_k = diffusion of turbulent kinetic energy
P_k = shear production of the turbulent kinetic energy
G_k = gravity production of turbulent kinetic energy
ε = turbulent energy dissipation

The eddy viscosity is calculated by

$$\nu_t = \sqrt{\overline{vv}}\, l_\mu \tag{3}$$

the ε is determined by

$$\varepsilon = \frac{\sqrt{\overline{vv}}\, k}{l_\varepsilon} \tag{4}$$

and l_μ, l_ε, and $\frac{\overline{vv}}{k}$ by the following equations respectively.

$$l_\mu = \frac{(0.33 + 0.214 f_{l_\mu})y}{1 + 5.025 \times 10^{-4} y_v^{*[1.53+0.12 f_{l_\mu}]}} \tag{5}$$

$$l_\varepsilon = \frac{(1.3 + 7.5 f_{l_\varepsilon})y}{1 + \left(2.12 + 7.88 f_{l_\varepsilon}\right)/ y_v^* + (0.028 + 0.0235 f_{l_\varepsilon}) y_v^*} \tag{6}$$

$$\frac{\overline{vv}}{k} = 0.4\left[1 - \exp\left(-\frac{y^{*(2-f_{vv/k})}}{4200(1 - 0.99 f_{vv/k})}\right)\right] \tag{7}$$

where

$$f_{l_\mu} = f_{l_\varepsilon} = \frac{1}{2}\left[1 + \tanh(50 * \left|Ar_y\right| - 4)\right] \tag{8}$$

$$f_{vv/k} = \frac{1}{2}\left[1 + \tanh(120 * \left|Ar_y\right| - 4)\right] \tag{9}$$

v = fluctuating velocity component in y direction

Since the standard k-ε model is well known, a reference to Launder and Spandling (1974) is sufficient.

There are no auxiliary flow and heat transfer models used. Otherwise, they should be reported here.

Boundary conditions

The boundary conditions for this room include flow obstacles of all solid objects, heat transfer from heated objects and walls, tracer gas sources, supply airflow, and exhausted flow.

The experimental data give the total heat flow from those heated objects that include the contributions from both convection and radiation. With this set of experimental data, neither a conjugate-heat transfer model with radiation nor a pure convective heat transfer model would be suitable for the heat transfer simulation. A feasible approach is to estimate the radiative heat transfer and to specify the convective heat flow boundary condition for the heated objects.

The radiative heat from a heated object is estimated as:

$$Q_{radiation} = \varepsilon_{object}\, \sigma\, (T^4_{object} - T^4_{walls})\, A_{object} \tag{10}$$

where ε_{object} = surface emissivity of the heated object
σ = Stefan-Boltzmann constant
T_{object} = surface temperature of the heated object

T_{walls} = surface temperature of the surrounding walls

A_{object} = surface area of the heated objects.

The surface temperatures of the heated objects, T_{object}, have to be estimated based on the user's experience, and thus uncertainties are introduced in the estimation. The convective heat is calculated as the total heat minus the radiative heat. The convective heat is assumed to be uniformly distributed on the entire surface of the heated objects. Obviously, it is questionable whether the uniform assumption is reasonable. This is another source of uncertainty.

Note that the estimation of the radiative heat could be considered as an auxiliary heat transfer model. However, it technically belongs to "boundary conditions", and is therefore presented in this section. The measured wall surface temperatures can be directly specified in the CFD code.

The tracer-gas source was set as a zero-momentum and zero-energy source. Considering the amount of total mass was very small, such an assumption seems reasonable.

The inlet conditions were assumed to have a uniform profile. Since the diffuser was a perforated surface with an effective area, the CFD simulation artificially increased the momentum for the velocity component normal to the wall by a factor of 1/(effective area ratio). This is an approximation method (Chen and Moser 1991) for simulating complex diffusers. This again introduced an error in the CFD simulation. The error is about 20% for the velocity near the diffuser.

The last component of the boundary conditions for this complete system (displacement ventilation in an office) is the exhaust. Zero pressure and zero gradient for all other flow parameters were used as boundary conditions for the exhaust.

Numerical methods

The fourth step of the reporting is the presentation of the numerical technique employed in the CFD analyses. The CFD code being used in this example discretized the differential equations by using the finite volume method with staggered grids. The CFD model was built in a Cartisian coordinate system, because the room geometry and the objects in the room were rectangular. The SIMPLE numerical algorithm was used. This numerical procedure is widely available in the literature. Hence, a reference to Patankar (1980) is sufficient. However, this CFD analysis defined a criterion of convergence as the maximum value of the absolute residuals of the transport equations U, V, W, P, T, k, ε, and C (SF_6 concentration) being less than 0.1% of the mass inflow times a reference value (the value at the inlet diffuser) of these variables. For example, mass inflow times supply air velocity can be used for U, V, and W. The CFD calculation used a linear relaxation factor of 0.8 for pressure and a false time-step of 0.1 s for all other flow parameters solved. A grid dependent study was also performed with three different grid resolutions: $29\times30\times19$, $48\times44\times24$, and $72\times66\times36$. For such a complicated system, it is very difficult to reach grid independent results. The results show that the difference between two finer grids is very small. Therefore, $48\times44\times24$ grids were considered to be sufficient. All this information should be reported in a CFD analysis of the indoor environment

Assessing CFD predictions

The next step of reporting CFD analysis in this example is the comparison of CFD results with the experimental data. This is the most important part of the reporting. The comparison can be as simple as smoke visualization that gives qualitative comparison. Figure 7 shows an example for such a comparison of the case shown in Figure 6. The graphical comparison should be accompanied by analysis of the results.

The comparison should also be quantitative. For example, Figure 8 presents the measured and computed velocity, temperature, SF_6 concentration, and velocity fluctuation in the center of office. The vertical axes are elevation in the room height. The horizontal axes are the parameters.

The validation should provide in-depth analysis on both CFD results and data quality. Figure 8, for example, illustrates that the velocity in most of the space (along the center of the office except near the floor), was lower than 10 fpm (0.05 m/s). The magnitude was so low that the hot-sphere anemometers may have failed to give accurate results. Nevertheless, the computed velocities agree well with the data. The velocity near the floor was larger than in the center of the room, because the diffuser was installed on the floor level. The temperature uncertainty in the measurement was 0.4°C. The CFD code clearly under-predicted the temperature, which was also found in the verification case, the mixed convection in the two-dimensional cavity. The discrepancy in this case is larger, because the boundary conditions were more complex. In addition, the uncertainty in estimating the convective heat transfer could have contributed to the larger error. The SF_6 concentration prediction is more dissatisfactory, compared with the velocity and temperature, especially in the upper part of the office. The reason may be due to the flow recirculation existing in the upper part of the office where the tracer-gas concentration was not uniform and very sensitive to the position and boundary conditions. The uncertainty and error for turbulent intensity is the largest in both the CFD model and the measurements. It is not surprising to see the large differences between the computed turbulent intensity and the measured data. In this case, it is difficult to judge whether the measured data or the computed results are more accurate.

Due to the limited space in this paper, the comparison of the computed results with the experimental data was done for one location. In practice, the comparison should be done in multi-locations.

Drawing conclusions

The final step in reporting the CFD results is to provide a sound conclusion. In this example, we may conclude that the CFD code is capable of simulating displacement ventilation in a room. The user can use the CFD code to design displacement ventilation system for the office shown in Figure 1. It is clear that the CFD code predicts mean flow parameters, such as air velocity, temperature, and contaminant concentration better than the second-order parameters, such as turbulence intensity. The user uses his/her knowledge extensively in verifying the code and validating CFD model. Therefore, the performance of the CFD code is also user-dependent.

A general discussion will be helpful to understand other important aspects of the work. For example, in this case, the user should point out that the SF_6 tracer gas can simulate not only a gaseous contaminant concentration such as carbon dioxide, but also aerosols like water vapor. This is very clear to people who know CFD but may not be so to others. The discussion should not be biased. For instance, the validation was only for displacement ventilation. One should not over-state the validity of the CFD results. In fact, the same CFD code and user combination has difficulty in predicting indoor environment with mixing diffusers (Chen and Srebric 2001).

In the validation case, all the wall temperatures were prescribed. In a real room as shown in Figure 1, the wall surface temperatures are unknown. The user could use an energy simulation program or a conjugate heat transfer model to determine the wall surface temperatures, with which further validation of the CFD simulation may be needed, and there may not be suitable experimental data for this validation. Therefore, the user can estimate the wall surface temperatures by some simple methods and report the uncertainties.

With the above-mentioned verification, validation, and reporting process, it is convincing that (1) the CFD code can be used to simulate the distributions of airflow, air temperature, relative humidity, turbulent intensity, and carbon dioxide concentration, and (2) the user is able to use the CFD code to conduct ventilation system design for the office shown in Figure 1.

This papers shows a reporting format for technical readers, because it provides rather detailed technical information. A report for non-technical reader can be much simpler, which may contains only sections "assessing CFD predictions" and "drawing conclusions".

DISCUSSION

The verification and validation cases used in this paper are with detailed experimental data. Very often, experimental data may not be available for a complete indoor environment system. This is especially true for industrial design and analysis. It is acceptable to utilize validations for several subsystems or a less-than-complete system. A subsystem of indoor environment represents some of the flow features in an indoor environment to be analyzed. The overall effect of several subsystems is equivalent to a complete system. For example, a complete indoor environment system consists of airflow and heat transfer in a room with occupants, furniture, and a forced air unit. If a user can correctly simulate several subsystems such as (1) airflow and heat transfer around a person, (2) airflow and heat transfer in a room with obstacles, and (3) airflow and heat transfer in a room with a forced air unit, the validation is acceptable. In the same example, a less-than-complete system for this environment can consist of airflow and heat transfer in a room with an occupant and a forced air unit. The furniture, although it affects the indoor environment, is not as important as the other components. Therefore, the validation with a less-than-complete system is acceptable. In either case, the key is that the validation should lead to a solid confirmation of the combined capabilities of the CFD user and code.

CONCLUSIONS

This paper illustrates how to use "The Manual for Verification, Validation, and Reporting of Indoor Environment CFD Analyses" (Chen and Srebric 2001 and 2002) by presenting an example of designing a displacement ventilation system in an office. The verification used a two-dimensional mixed convection case that has the same basic flow features as the office. With the excellent quality of the experimental data, it is possible to verify the ability of the CFD code for simulating the mixed convection flow in a room.

Then, a three-dimensional an environmental chamber with displacement ventilation was used for validation. The chamber represents the complete flow, heat transfer, and mass transfer features as those in the office where displacement ventilation is to be designed. The validation procedure demonstrates the importance of the CFD user in the simulation, because many engineering assumptions should be made. An accurate and successful simulation is a combined effort of the CFD code and the CFD user. After the validation, the CFD user is capable to design the ventilation system in the office.

Although the reporting of CFD analyses does not have to be in a uniform format, it generally contains the following aspects for technical readers: experimental design, turbulence model and auxiliary heat transfer and flow models, boundary conditions, numerical methods, assessing CFD predictions, and drawing conclusions.

REFERENCES

Baker, A.J. and Kelso, R.M., 1990. “On validation of computational fluid dynamics procedures for room air motion prediction,” *ASHRAE Transactions*, 96.(1), 760-774.

Baker, A.J. and Gordon, E.B., 1997. “Computational fluid dynamics - a two-edged sword,” ASHRAE Journal, 39(8), 51-58.

Blay, D., Mergui, S., and Niculae, C. 1992. “Confined turbulent mixed convection in the presence of a horizontal buoyant wall jet,” *Fundamentals of Mixed Convection,* ASME HTD-Vol. 213, pp. 65-72.

Chen, Q. and Moser, A. 1991. “Simulation of a multiple-nozzle diffuser,” *Proc. of the 12th AIVC Conference on Air Movement and Ventilation Control within Buildings*, Ottawa, Canada, Vol. 2, pp. 1-13.

Chen, Q. and Srebric, J. 2001. “How to verify, validate, and report indoor environment modeling CFD analyses,” Final Report for ASHRAE RP-1133, 58 pages, Welsh School of Architecture, Cardiff University, UK and Department of Architectural Engineering, Pennsylvania State University, PA.

Chen, Q. and Srebric, J. 2002. "A procedure for verification, validation, and reporting of indoor environment CFD analyses," Accepted by International Journal of HVAC&R Research.

Fanger, P.O., Melikov, A.K., Hanzawa, H., and Ring, J. 1989. “Turbulence and draft,” *ASHRAE J*ournal, 31(7), 18-23.

ISO.1990. “Moderate thermal environments-determination of the PMV and PPD indices and specification of the conditions for thermal comfort,” ISO standard 7730, International Standards Organization, Geneva.

Launder, B.E., and Spalding, D.B. 1974. “The numerical computation of turbulent flows,” In *Computer Methods in Applied Mechanics and Energy*, Vol. 3, pp. 269-289.

Murakami, S., Kato, S., and Zeng, J., 1997. “Flow and temperature fields around human body with various room air distribution -- CFD study on computation thermal manikin – Part I,” *ASHRAE Transactions*,103(1), 3-15.

Murakami, S., Kato, S., and Zeng, J., 1998, “Numerical simulation of contaminant distribution around a modeled human body: CFD study on computation thermal manikin – Part II,” *ASHRAE Transactions*, 104(2).
Nielson, P.V. 1974. "Flow in air conditioned rooms," Ph.D. Thesis, Technical University of Denmark, Copenhagen.
Nielsen, P.V. 1995, “Airflow in a world exposition pavilion studied by scale-model experiments and computational fluid dynamics,” *ASHRAE Transactions*,101(2), 1118-1126.
Patankar, S.V. 1980. *Numerical Heat Transfer and Fluid Flow*, Hemisphere Publishing Corporation.
ROOMVENT 2000, Proceedings of the 7th International Conference on air distribution in rooms, Reading, UK.
Xu, W. and Chen, Q. 2000. “Simulation of mixed convection flow in a room with a two-layer turbulence model,” *Indoor Air*, 10, 306-314.
Yuan, X., Chen, Q., Glicksman, L.R., Hu, Y., and X. Yang. 1999. “Measurements and computations of room airflow with displacement ventilation,” *ASHRAE Transactions*, 105(1), 340-352.

Table 1. Relaxation method and factors used in Blay’s case.

	P	U	V	T	k	ε
Relaxation method	Linear relaxation	False-time-step	False-time-step	False-time-step	False-time-step	False-time-step
Factor	0.8	1.44	1.44	8.64	0.288	0.288

Table 2. Detailed thermal boundary conditions for the heated objects in the office.

Item	Length	width	height	location			heat
	Δx [m]	Δy [m]	Δz [m]	x [m]	y [m]	z [m]	Q [W]
Room	5.16	3.65	2.43	0.0	0.0	0.0	
Window	0.02	3.35	1.16	5.16	0.15	0.94	
Diffuser	0.28	0.53	1.11	0.0	1.51	0.03	
Exhaust	0.43	0.43	0.0	2.365	1.61	2.43	
Occupant1	0.4	0.35	1.1	1.98	0.85	0.0	75
Occupant2	0.4	0.35	1.1	3.13	2.45	0.0	75
Computer1	0.4	0.4	0.4	1.98	0.1	0.75	108.5
Computer2	0.4	0.4	0.4	3.13	3.15	0.75	173.4
Table1	2.23	0.75	0.01	0.35	0.0	0.74	0.0
Table2	2.23	0.75	0.01	2.93	2.90	0.74	0.0
Box1	0.33	0.58	1.32	0.0	0.0	0.0	0.0
Box2	0.95	0.58	1.24	4.21	0.0	0.0	0.0
Lamp1	0.2	1.2	0.15	1.03	0.16	2.18	34
Lamp2	0.2	1.2	0.15	2.33	0.16	2.18	34
Lamp3	0.2	1.2	0.15	3.61	0.16	2.18	34
Lamp4	0.2	1.2	0.15	1.03	2.29	2.18	34
Lamp5	0.2	1.2	0.15	2.33	2.29	2.18	34
Lamp6	0.2	1.2	0.15	3.61	2.29	2.18	34

Note: 1. x is from west to east, y from south to north, z from low to high.
2. The coordinates of the item in the table are the south-west-low corner of the item.
3. The heat generated includes radiation and convection.
4. The effective area ratio of the diffuser is 10%.

Figure 1. The configuration of an office where displacement ventilation is to be designed.

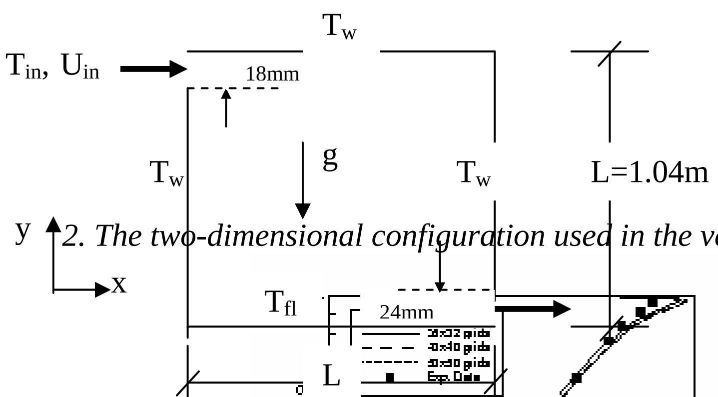


Figure 2. The two-dimensional configuration used in the verification calculation.

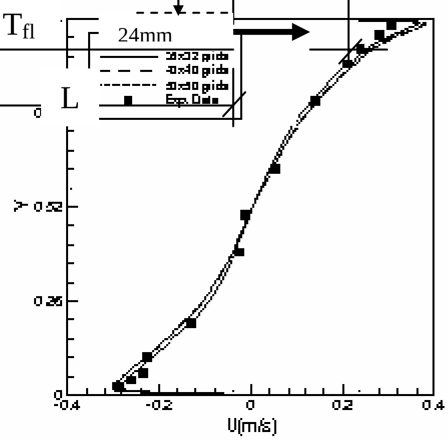
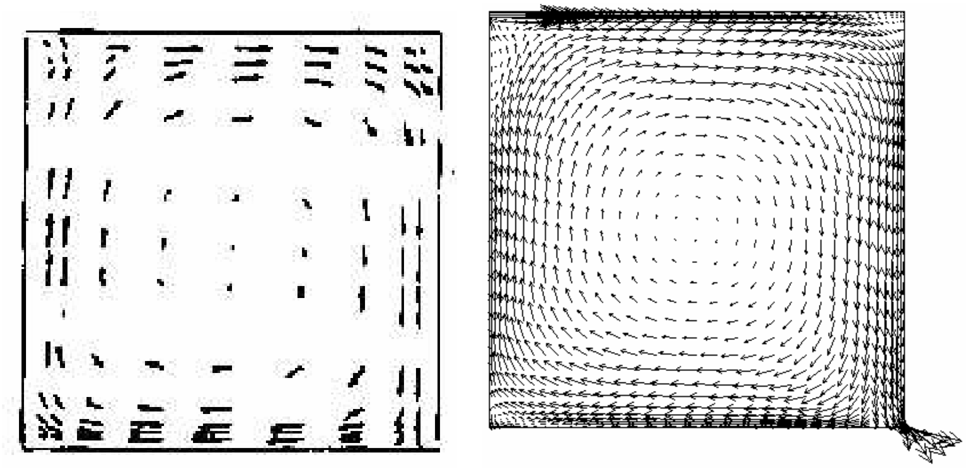


Figure 3. Grid-dependent study: mean velocity comparison at x/L=0.5 with data from Blay et al. 1992).



(a) (b)

Figure 4. Mixed convection in a cavity with Fr=5.31. (a) Observed flow pattern and (b) predicted flow pattern.

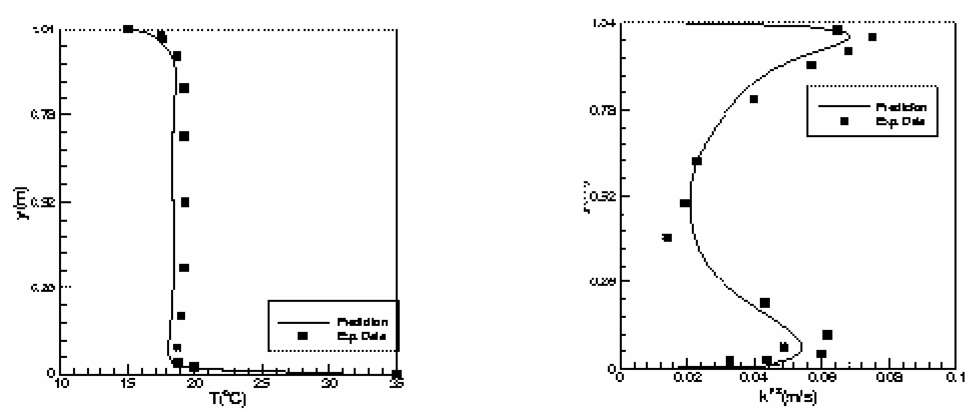


Figure 5. Validation of the computed mean temperature and turbulence kinetic energy with the experimental data.

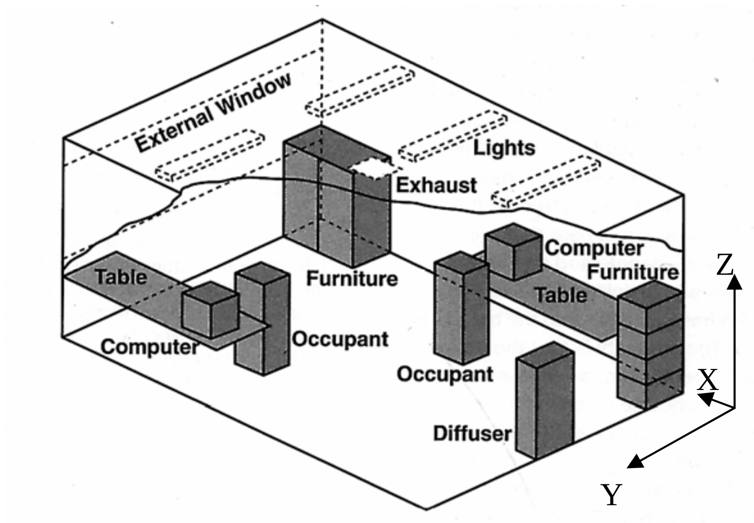


Figure 6. The office configuration used in the model.

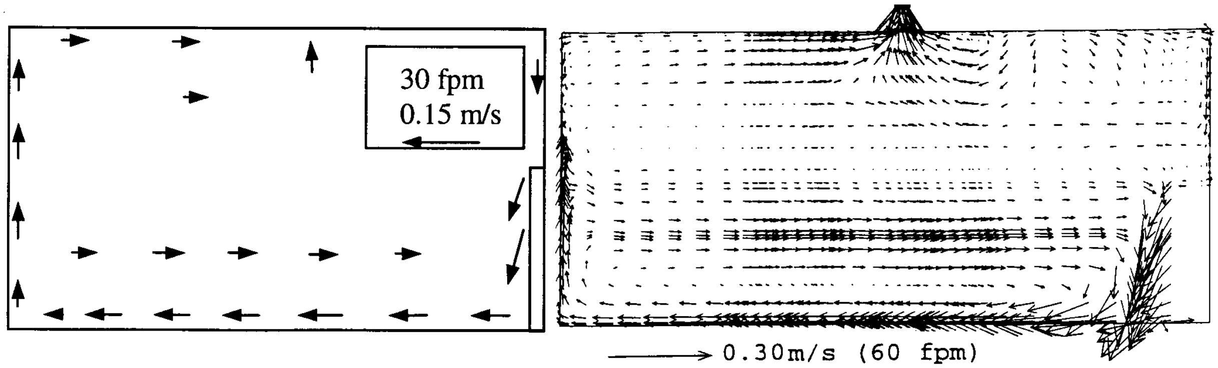


Figure 7. The airflow pattern observed by using smoke visualization (left figure) and computed by the CFD program (right figure) in the mid-section (The length of the arrows is proportional to the velocity magnitude).

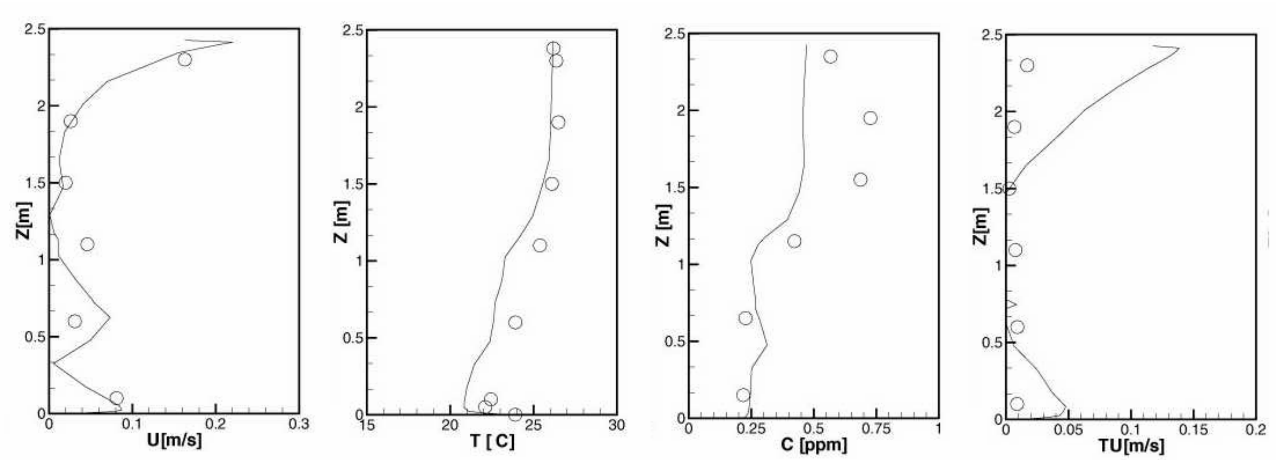


Figure 8. The comparison between the CFD results (lines) and experimental data (circles) at the center of the office. (a) Velocity, (b) temperature, (c) SF_6 concentration, and (d) turbulent intensity.