

APPLICATION OF CFD FOR TEMPERATURE AND AIR VELOCITY DISTRIBUTION CALCULATION IN A VENTILATED ROOM

ZYGMUNT WIERCIŃSKI^{1,2}
AND ALDONA SKOTNICKA-SIEPSIAK²

¹*Institute of Fluid-Flow Machinery PAS,
Fiszera 14, 80-952 Gdansk, Poland
zw@imp.gda.pl*

²*University of Warmia and Mazury,
Oczapowskiego 2, 10-719 Olsztyn, Poland*

(Received 16 May 2008; revised manuscript received 17 July 2008)

Abstract: Appropriate air distribution in a room is necessary for thermal comfort. By using Computational Fluid Dynamics (CFD) it is possible to compare optional ways of air supply and distribution at the stage of the ventilation design concept. Using these simulations the ventilation system designer can choose the best method of air supply in the room diminishing the risk of an incorrect solution.

Keywords: air distribution, ventilation, CFD

1. Introduction

Room air distribution shows how air is introduced into the room, how it flows through the room and how it is removed from the room. Despite a correct estimation of the air quantity, temperature and humidity, the correct matching of the pressure and volume rate of the fan and the correct dimensioning of ducts, thermal comfort will not be achieved in the ventilated space when the room air distribution is incorrect. The location of and interaction between air outlets and inlets are of fundamental importance for the distribution of air in space. Many other factors have also impact on the thermal comfort in a room, for example, the physical activity of its occupants, the temperature of the air and the walls, the humidity and velocity of the air, the cleanness of air, the noise and many others. To keep thermal comfort in a room we must know and control at least the temperature of air, its humidity, velocity and the temperature of the walls [1].

Each room requires an individual calculation of factors which have impact on the thermal comfort and a careful analysis of the air supply and distribution in the ventilated space.

It is well known that three basic methods of air flow organization in a room can be identified:

1. top-bottom – this is the most frequently used airflow organization method. Particular attention should be paid to the neighborhood of the ceiling which influences the air stream flow. The selection of air inlets is related to the necessity of analyzing the range of the air stream, velocity and temperature in the final stage where the air stream approaches the zone occupied by people;
2. bottom-top – usually applied to rooms with air heating. It requires maintaining a relatively low air velocity because of the possibility of raising particles of dust from the floor;
3. top-top – with an appropriate selection of inlets, it ensures effects comparable with the top-bottom organization of flow. It is frequently applied in the case of upgrading the ventilation systems in rooms where it is possible to install suspended ceilings.

It is only the CFD simulation that provides the designer with a tool to choose the air supply method and – to some extent – to check it before installing a ventilation system. In the design practice, there are only some tips, general cases and a limited number of examples for some spaces. The designer must decide about the air supply and distribution on the basis of his/her own intuition. A wrong decision can cause incorrect room air distribution, and in consequence, a failure to provide thermal comfort in the ventilated space. Often, once an incorrect design has been implemented, it is not possible to improve the air distribution or it can result in excessive costs, very often the change is restricted by the technology. It is only the use of the CFD simulation that gives an opportunity of checking optional designs and selecting an optimal solution.

The FloVent software developed by Flomerics, Great Britain [2], was used for our air distribution simulations. The PC, (Pentium 4, 2.8GHz, 1GB memory) needed about 150 minutes for calculation when the analyzed space was divided into 800 000 cells, and about 80 minutes when 160 000 cells were used in the calculation.

2. FloVent Package and Computational Fluid Dynamics (CFD) outline for ventilation problems

The FloVent package enables a CFD simulation which relies on the numerical calculation of the fluid flow, heat transfer and related processes such as radiation. The purpose of the FloVent package is to provide the engineer with a computer-based tool giving a chance for analyzing air-flow processes in and around buildings with the aim of improving and optimizing the design of new or existing heating or ventilation systems.

In CFD, the mathematical simulation of a fluid flow and heat transfer involves the solution of a set of coupled, non-linear, second order, partial differential equations usually called Navier-Stokes equations and their simplified form namely Reynolds equations. These differential equations that these field variables satisfy are transformed as equations of conservation. These equations are written below:

– continuity equation:

$$\operatorname{div}U = 0, \tag{1}$$

– momentum equation (Navier-Stokes equation):

$$\frac{dU}{dt} = g - \frac{1}{\rho} \operatorname{grad}p + \operatorname{div}(2\nu E), \tag{2}$$

– energy balance equation:

$$\rho c_v \frac{dT}{dt} = \operatorname{div}(\lambda \operatorname{grad}T) + \dot{D} + \dot{\sigma}, \tag{3}$$

where:

- U – velocity vector of the air with components: U_1, U_2, U_3 ,
- g – mass forces vector,
- p – pressure,
- ρ – density,
- ν – kinematic viscosity,
- E – deformation rate tensor,
- T – air temperature,
- c_v – constant volume specific heat,
- λ – thermal conductivity coefficient $\lambda = \lambda(p, T)$,
- \dot{D} – kinetic energy dissipation intensity,
- $\dot{\sigma}$ – intensity of internal heat sources in the unit of volume.

In the FloVent package the finite-volume method is used to solve this set of equations. These equations are solved for each grid cell (Figure 1).

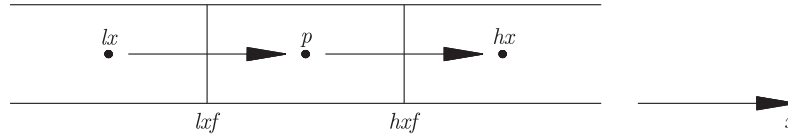


Figure 1. Diagram of a single computation element (cell)

The mass balance for an element is given by the following equation:

$$\left(\frac{\rho_p - \rho_t}{\delta t} \right) V_p + (\rho u)_{h_{xf}} A_x - (\rho u)_{l_{xf}} A_x = 0. \tag{4}$$

The mass flow rate in the element and its difference in the stream between the cell surfaces will be zero. The finite-volume equation method is also used to calculate periodical conductivity according to the following equation:

$$\int_z \int_y \int_x \frac{-\partial}{\partial x} \lambda \left(\frac{\partial T}{\partial x} \right) dx dy dz = - \left[\lambda \left(\frac{T_{hx} - T_p}{\delta x} \right) - \lambda \left(\frac{T_p - T_{lx}}{\delta x} \right) \right]. \tag{5}$$

The differential equations problem, such as the flow in ventilation, requires boundary conditions. Of course, the air velocity near the body surface must be equal to zero – there is no slip, so the most important condition is that the air velocity near the body surface equals zero. In CFD models the problem of friction at a smooth surface is described by the following formulas (see also Figure 2):

– friction velocity:

$$u_\tau = \sqrt{\frac{\tau_w}{\rho}}, \quad (6)$$

– the local Reynolds number:

$$y^+ = \frac{u_\tau y \rho}{\mu}, \quad (7)$$

– when $y^+ > 11.5$ then (turbulent):

$$\frac{u}{u_\tau} = \frac{1}{0.435} \ln(9y^+), \quad (8)$$

– when $y^+ \leq 11.5$ then (laminar):

$$\frac{u}{u_\tau} = y^+. \quad (9)$$

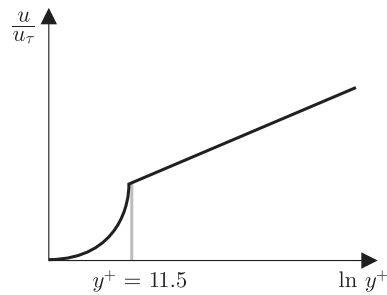


Figure 2. Boundary layer structure with laminar and turbulent sub-layer in wall co-ordinates

For calculation of the flow at the rough wall the roughness height h in the Reynolds number should be involved:

$$\text{Re}_r = \frac{h u_\tau \rho}{\mu}. \quad (10)$$

Generally, the calculation involves the case of a smooth surface, and only for the condition when $\text{Re}_r > 3.3$, the above formula is used for the determination of u_τ and next the friction stress τ_w at the surface and further the velocity at the wall layer is calculated as follows:

$$\frac{u}{u_\tau} = \frac{1}{0.435} \ln\left(\frac{29.7y}{h}\right). \quad (11)$$

The modeling of turbulence poses a separate problem because usually flows in ventilated rooms are turbulent, and it is rather rarely that a laminar flow has to be considered except perhaps for the well known ventilation laminar ceilings.

Nowadays, computers cannot solve complete Navier-Stokes equations. To enable a solution of the Navier-Stokes equations, each variable appearing in the equations

should be decomposed into two parts: time average and fluctuating components, like in the formula below:

$$U = \bar{U} + u', \quad h = \bar{h} + h'. \quad (12)$$

Introducing these and similar equations into Navier-Stokes equations, we obtain the Reynolds equations, and firstly the momentum equation can be written as:

$$\frac{\partial}{\partial x_j} (\rho U_i U_j) = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left(\mu \frac{\partial \bar{U}_i}{\partial x_j} - \rho \bar{u}'_i \bar{u}'_j \right) + g(\rho - \rho_0) \quad (13)$$

and next the enthalpy equation has the form:

$$\frac{\partial}{\partial x_j} (\rho \bar{U}_i \bar{h}) = \frac{\partial}{\partial x_j} \left(\lambda \frac{\partial \bar{T}}{\partial x_i} - \rho \bar{u}'_i \bar{h}'_j \right). \quad (14)$$

The natural convection can be important and should be taken into account in the ventilation problem. Convection is caused by gravity and density (temperature) difference, and in the FloVent package, the air properties can be treated either as fixed or variable. When the air density is constant, the last part of Equation (13) will be given by Boussinesq approximation:

$$g(\rho - \rho_0) = g\rho \frac{T - T_{\text{ref}}}{T_e}, \quad (15)$$

where $g = 9.81 \text{ m/s}^2$ is the gravity acceleration, $T_{\text{ref}} = 20^\circ\text{C}$, $T_e = 300 \text{ K}$ corresponds to the default temperature for determination of the air volume thermal coefficient $\beta = 1/T_e$. When the density is variable, the last part of Equation (13) remains unchanged.

In Equations (13)–(14) the extra components reveal: the turbulent stress and the turbulent enthalpy flow rate:

$$S_t = -\rho \bar{u}'_i \bar{u}'_j, \quad (16)$$

$$h_t = -\rho \bar{u}'_i \bar{h}'_j. \quad (17)$$

As a consequence, what is well known, there are more unknowns than equations. In that case we need modeling of these new unknowns.

Thus, the turbulent stresses can be determined from the following equation:

$$-\rho \bar{u}'_i \bar{u}'_j = \mu \left(\frac{\partial \bar{U}_i}{\partial x_j} - \frac{\partial \bar{U}_j}{\partial x_i} \right) - \frac{2}{3} \rho k \delta_{ij}, \quad (18)$$

where δ_{ij} is the Kronecker's delta, k is the kinetic energy of turbulence.

In computations with the FloVent package three options can be chosen for turbulence modeling:

1. Invariable viscosity, where viscosity and turbulence conductivity are determined by the equations:

$$\nu_t = C\nu_l, \quad \lambda_t = \frac{\nu_t C_p}{\text{Pr}_t}, \quad (19)$$

ν_t – turbulent viscosity, ν_l – laminar viscosity, constant $C = 100$ and $\text{Pr}_t = 0.9$.

This model represents some extra stirring caused by the turbulence and local effects caused by the surface without an extra equation for transport. It is a default model, which is used for starting calculations and quick receiving of preliminary results.

2. Model LVEL [3] is a simple algebraic model which does not need an estimation of the turbulent viscosity. In that model it is needed to determine the distance to the closest wall, the local speed and the local, effective viscosity by using the molecular viscosity. The distance is determined by solving the Poisson equation within the analyzed 3D system:

$$\nabla\phi = -1, \tag{20}$$

where $\phi = 0$ for the wall.

Thanks to that equation the maximum length scale D and local length L from the nearest wall is to be estimated:

$$D = \sqrt{(|\nabla\phi|^2 + 2\phi)}, \tag{21}$$

$$L = D - |\nabla\phi|. \tag{22}$$

Next, based on local velocity U and length scale L , the Reynolds number can be determined. By using universal law of wall's dimensionless effective viscosity can be calculated by differentiating of the wall function:

$$\nu^+ = \frac{dy^+}{du^+}. \tag{23}$$

In that way, it is possible to estimate the effective viscosity in each cell of the flow.

Another algebraic model which can be used in the ventilation calculation is the so called LVEL CAP model. It is a slightly changed LVEL model. The L value is calculated by the LVEL model but velocity U is determined by the user. The viscosity is estimated according to the LVEL model and according to the equation:

$$\mu_t = \alpha\rho\bar{U}\bar{L}, \tag{24}$$

where $\alpha = 0.01$. In this model, however, certain caps related to a higher limit for the velocity determined by the user have been imposed.

3. Two equation k - ε turbulence model.

In that model, according to the equation:

$$\mu_t = f(k, \varepsilon), \tag{25}$$

the turbulent viscosity depends on two variables: the kinetic energy of turbulence k and the dissipation rate ε of kinetic energy k .

In a typical case the eddy viscosity is determined by the equation:

$$\mu_t = C_\mu\rho\frac{k^2}{\varepsilon}. \tag{26}$$

The corresponding transport equations for the values of turbulent energy k and its dissipation rate ε for all computation cells except for the wall layer are given below:

$$\frac{\partial\rho\bar{U}_i k}{\partial x_i} = \frac{\partial}{\partial x_i} \left(\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_i} \right) + P + G - \rho\varepsilon, \tag{27}$$

$$\frac{\partial\rho\bar{U}_i \varepsilon}{\partial x_i} = \frac{\partial}{\partial x_i} \left(\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_i} \right) + C_1 \frac{\varepsilon}{k} (P + C_3 G) - C_2 \rho \frac{\varepsilon^2}{k}, \tag{28}$$

where P is the production of friction stress:

$$P = \mu_{\text{ef}} \frac{\partial \bar{U}_i}{\partial x_j} \left(\frac{\partial \bar{U}_i}{\partial x_j} + \frac{\partial \bar{U}_j}{\partial x_i} \right), \quad (29)$$

while G is the production of turbulence kinetic energy caused by uplift pressure:

$$G = \frac{\mu_{\text{ef}}}{\sigma_T} \beta g_i \frac{\partial T}{\partial x_i}. \quad (30)$$

In a typical version of this model the values of the constants are as follows:

$$C_\mu = 0.09, C_1 = 1.44, C_2 = 1.92, C_3 = 1.0, \sigma_k = 1.0, \sigma_\varepsilon = 1.217. \quad (31)$$

The great advantage of this model is its simplicity and stability.

The turbulent diffusivity is related to the turbulent viscosity by the following equation:

$$D_t = \frac{\mu_t}{\rho \text{Sc}_t}, \quad (32)$$

where D_t is the coefficient of diffusion and Sc_t is the turbulent Schmidt number with the assumption that the value of turbulent Schmidt number is $\text{Sc}_t = 1.0$.

In our calculations the k - ε model has been used without taking into account the wall roughness, so the case of a smooth wall has been considered.

3. Simulation results

The modernization of the ventilation system in a lecture hall in one of the buildings of the Warmia and Mazury University in Olsztyn has been used as a calculation example. Computations of the air distribution using the CFD for two alternative designs of the positioning of ventilation inlets and outlets in a small lecture hall are the case studies analyzed. The computations shown below were carried out for a lecture hall for 60 persons that was planned for modernization, including air conditioning system installation. Due to a large height of the hall, it was decided to apply a suspended ceiling and to install an air conditioning unit on the roof of the building, as the hall is situated on the top floor of the building. For those reasons it was decided to run the inlet and outlet air ducts in the space between the suspended and the actual ceiling, *i.e.* to organize the air circulation according to the top-top system verifying at the same time two different configurations of air inlets and outlets.

The data for computations of heat and humidity losses and gains were assumed on the basis of the data for the climatic zones: summer (zone II) and winter (zone IV) for the city of Olsztyn.

The heat and humidity gains were calculated in the room intended for not more than 60 people and the air volume delivered to this room amounted to 2500 m³/h according to this calculation. Additionally, two different displacements of inlet and output vents were considered, both as vents in the suspending ceiling, named below as case A and B.

- A. In the first case of air supply, two rows of square diffusers in the ceiling along the room are mounted which bring air into the space. Outlet square diffusers are placed between them in the third row in the ceiling.
- B. In the second room air supply case, the square diffusers which supply air into the room are mounted in the central part of the ceiling. The outlet square

diffusers are placed outside the inlet diffusers, so there are two rows of three outlet diffusers at the end and at the beginning of the room.

Analyzing the results of the velocity and temperature distribution simulation for the above two cases it can be said that the case denoted as B is better than case A of the air distribution organization. Whereas the temperature distribution in both cases is good enough to preserve the thermal comfort condition, the air velocity distribution in case A is rather better where the velocity amplitudes are lower than in case B.

Firstly, in both cases, A and B, quite good temperature distributions in the space can be seen, Figures 3 and 4. In case A, an increase in the temperature in the central part of the room is provided, above the place where the largest concentration of people is assumed, Figure 3. For case B of the room air supply, a temperature increase in the front part of the lecture hall is shown, Figure 4. However, in both cases the temperature does not exceed 22.5°C . For both cases, A and B, the section at the height of 3.5 m is probably the place where the highest air temperature can be seen because of the expected temperature gradient from the floor to the ceiling. The lowest temperature is also seen in these pictures, Figures 3 and 4, this is placed at the inlet vents and it is equal to 18°C so is equal to the design temperature of the inlet air. In these figures, a different displacement of vents according to case A or B is can be easily seen, as well.

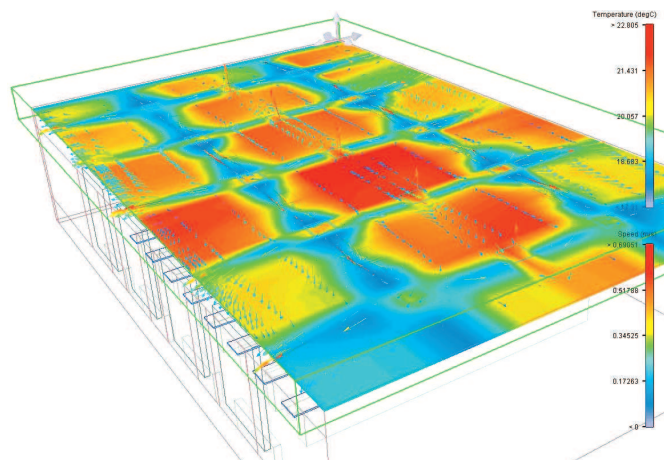


Figure 3. Temperature distribution and air velocity for case A of air supply 3.5 m from the floor

Analyzing the air velocity distribution, Figures 5 and 6, it is possible to say that the air velocity distribution in case A is more balanced than in case B.

In Figure 5 the air velocity distribution in case A is shown for the coordinate $Z = 5 \text{ mm}$, *i.e.* in the middle section of the room. The velocity differences on the diagram are small showing almost no vortices, the highest velocity differences are seen close to the vents at the ceiling.

In case B, a whirl in the central part of the room is observed which can cause an unpleasant feeling of draught. Unfortunately, in the second air distribution case,

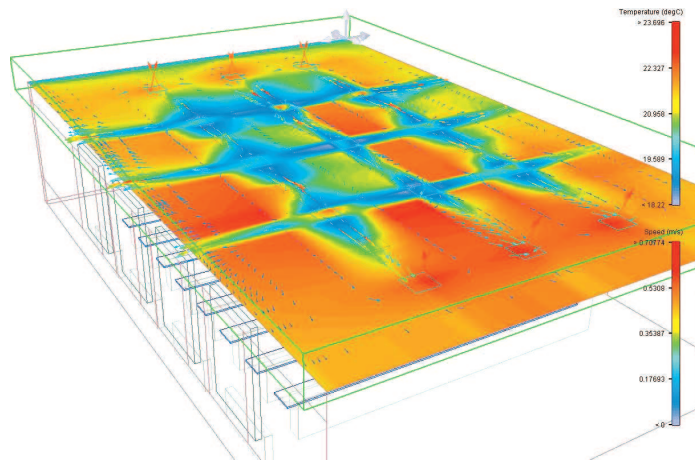


Figure 4. Temperature distribution and air velocity for the case B of the air supply in 3.5 m from floor

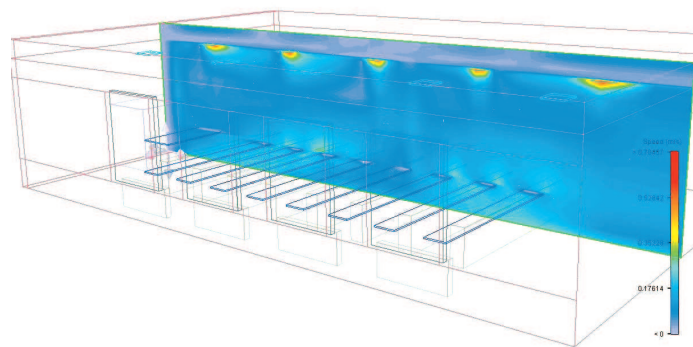


Figure 5. Air velocity distribution in the case A at the section $Z = 5$ m

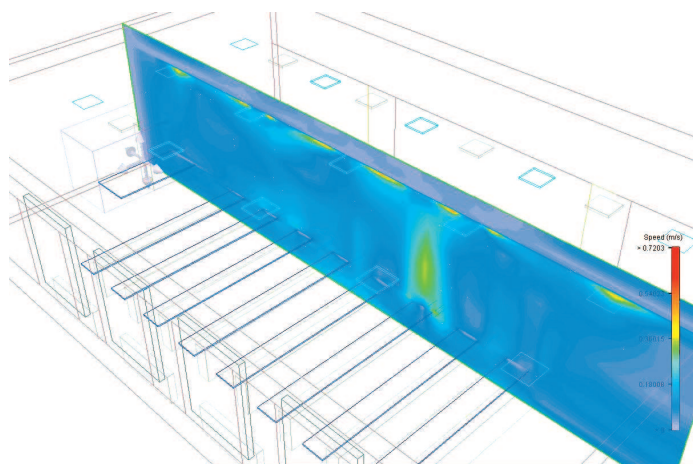


Figure 6. Air velocity distribution in section $Z = 5$ m for option B

we can find a vortex in the central part of the space. In that case we do not have any thermal control in the room.

4. Conclusions

The CFD package is used for an air distribution simulation in the ventilated room which enables an analysis of different vent displacements. This analysis can be used to facilitate the easier choice and proper design of the positioning of vents in order to guarantee thermal comfort in the ventilated room. Nowadays, when the HVAC systems are much often applied in the building environment it can be used as a proper tool for designers in their work. It cannot be forgotten that also a CFD simulation has its limitations resulting from imperfect turbulence modeling and a limited number of cells in the considered space. The graphical results of the simulation can also enhance the contact between the designer and the investor.

References

- [1] Recknagel I H, Sprenger E, Hönnmann W and Schramek E 1994 *Taschenbuch für Heizung und Klimatechnik*, Oldenbourg Verlag, München
- [2] Flomerics Ltd 2003 *FloVent Manual Set*, Ver. 7
- [3] Agonafer D, Gan-Li L and Spalding D B 1996 *Appl. CAE/CAD Electronic Systems*, ASME EEP, **18**, pp. 23–26