Numerical modelling of air supply and air flow pattern of a room that contains a gas appliance with open combustion chamber

Dr. Lajos Barna PhD and Róbert Goda

Budapest University of Technology and Economics Department of Building Service Engineering, Hungary

Corresponding email: tanszek@epgep.bme.hu

SUMMARY

In the last decade, quite a few carbon-monoxide intoxications occurred in Hungary due to the inadequate operation of gas appliances with open combustion chamber, connected to chimneys. These cases emphasized the importance of faultless air supply of the appliances and the safe removal of the incipient flue gases. This problem gave reason for the modelling of air supply, temperature and velocity distributions in the space that contains the gas appliance. For the modelling of changes caused by the variations in the inside or outside ambient conditions, numerical modelling can be used. With the help of CFD, the phenomena can be studied in what is virtually a computational environment. CFD modelling gives results for the changes in the magnitude and direction of air velocity in the room and between the air inlet and the appliance, temperature distribution in the room and from the weather factors, the effect of wind on the operation of the air inlet.

INTRODUCTION

“B” type gas appliances have an open combustion chamber; combustion air comes from the room in which the equipment operates, while flue gases leave through a chimney. The two primary groups of “B” type gas appliances according to the European grouping are [8]:

– Appliances with atmospheric burner and draught hood, connected to a chimney with natural draught (e.g. B11, Figure 1.),
– Appliances, which have burners installed with ventilators, connected directly to the chimney, without draught hood (e.g. B23, B33, B53).

Figure 1. B11 type gas appliances, with opened combustion chamber, connected to a chimney
9-10 million gas appliances are estimated to operate in Hungary, most of which connected to a chimney and have open combustion chamber (B_{11} type gas appliances). In the case of these appliances, flue gas has immediate contact with the air of the room in which the machine is installed. Thus, if the air-flow conditions are unfavourable, the flue gas may re-enter the space.

Air flows into the room due to the draught in the chimney. To be able to determine the operating point of the system the mathematical model of the chimney, the gas appliance, the room and the air inlet has to be worked out. However, the simultaneous or variance-based examination of several factors cannot be carried out analytically because of the large number of equations and their complexity (differential and integral equations etc.). For the modelling of changes caused by the variations in the inside or outside ambient conditions, numerical modelling can be used. With the help of CFD, the phenomena can be studied in what is virtually a computational environment.

CFD modelling gives results for the changes in the magnitude and direction of air velocity in the room and between the air inlet and the appliance, temperature distribution in the room and from the weather factors, the effect of wind on the operation of the air inlet.

METHODS

Steps of the CFD modelling:

– creating the geometry of the model,
– stating the differential equations for the numeric model,
– developing the CFD model,
– modelling the air supply and flue gas removal of a gas appliance for different conditions and operation modes, compute the air velocity and temperature distribution in the room.

First step of investigations: creating the geometric model

For the modelling of the B_{11} type gas appliance a conventionally sized room is used, in which the appliance is the only equipment (Figure 2). A volume of the room is 15 m³, and its size in detail is: 2 m (width), 2.5 m (length), 3 m (height). The windows and doors of the room are air-tight structures made of wood or plastic, sealed with several layers of rubber sealing. Outside air can barely or cannot enter at all in the room through natural (gravitational) means. The air necessary for combustion is provided via air inlets that in the model were inserted in the window frame in different ways, under and above the window.

Figure 2. The geometric model used for the examination of B_{11} type gas appliances
1 – wall-mounted gas appliances, 2 – window, 3 – radiator, 4 – air inlet, 5 – chimney
The $U$-value of the external wall is 0.45 W/m$^2$·K, while the window has a $U$-value of 1.4 W/m$^2$·K.

Under the window a radiator is situated that is controlled by a thermostatic radiator valve which adjusts the heat loss so that the desired room temperature is achieved.

Nominal heat output of the gas appliances in the investigation are: 12 kW, 24 kW, 28 kW and 36 kW.

The connecting flue pipe consist of: 0.5 m long vertical section, bend, 1 m long horizontal section. The pipe is made of aluminium and has a maximum absolute roughness of 1 mm.

The chimney is situated partly in the heated space and partly outside.

The outdoor air temperature is -15 °C, which is the best condition regarding the chimney but is the worst from the room’s comfort point of view.

Figure 3 shows the main sizes of the geometric model that was shown in Figure 2.

![Figure 3. The main sizes of the geometric model](image)

**Stating the differential equations for the numeric model**

The numeric model, based on the geometric model, was developed by adding principal initial and boundary conditions.

The air movements of closed areas are described by the differential equations of continuity and *Navier-Stokes*. The thermo balance of the areas is expressed by the equation of energy; its distribution of concentration is described by the differential equation of material balance. As we are talking about turbulent air conduction, also the proportion of the kinetic energy and the dissipation ($k$-$\varepsilon$) of the airflow has to be determined. Resulting from a system of equations, this is the mathematical model of closed spaces.

Assuming an incompressible agent the listed equations are formed as follows:

**Continuity:**

\[ \text{div} \ (\rho \cdot u_i) = 0 , \quad (1) \]

where $\rho$ is the air density and $u_i$ are the air velocity components in $x$, $y$, $z$ direction.
Equation of movement:

\[
\frac{\partial}{\partial x_i} (\rho \cdot u_i \cdot u_j) = \frac{\partial}{\partial x_i} \left( (\mu + \mu_t) \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right) - \frac{\partial}{\partial x_j} \left( p + \frac{2}{3} \rho \cdot k \cdot \delta_{ij} \right) + g_i (\rho_x - \rho),
\]

(2)

where \( \mu \) is the viscosity, and \( \mu_t \) is the turbulent viscosity, \( p \) is the pressure, \( k \) is the kinetic energy, and \( \delta_{ij} \) is the Kronecker symbol.

Equation of energy:

\[
\frac{\partial}{\partial x_i} (\rho \cdot u_i \cdot h) = \frac{\partial}{\partial x_i} \left( \frac{\mu}{\sigma} + \frac{\mu_t}{\sigma_t} \right) \frac{\partial h}{\partial x_i} + Q,
\]

(3)

where \( h \) is the enthalpy, \( Q \) is the quantity of heat per volume, \( \sigma_t \) is a factor, depends on Prandtl- and Schmidt-numbers.

The turbulent viscosity:

\[
\mu_t = K \cdot \rho \cdot \frac{k^2}{\varepsilon},
\]

(4)

where \( K \) is a constant, \( \varepsilon \) is the dissipation of kinetic energy.

Turbulent kinetic energy:

\[
\frac{\partial}{\partial x_i} (\rho \cdot u_i \cdot k) = \frac{\partial}{\partial x_i} \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_i} - K_A \cdot \rho \cdot \varepsilon + \mu_t \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) + F
\]

(5)

where \( \sigma_k \) is the kinetic energy factor.

Dissipation of turbulent kinetic energy:

\[
\frac{\partial}{\partial x_i} (\rho \cdot u_i \cdot \varepsilon) = \frac{\partial}{\partial x_i} \left( \mu + \frac{\mu_t}{\sigma_E} \right) \frac{\partial \varepsilon}{\partial x_i} - K_2 \cdot \rho \cdot \frac{\varepsilon^2}{k} + K_1 \cdot \mu_t \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\varepsilon}{k} + K_3 \cdot F \cdot \frac{\varepsilon}{k},
\]

(6)

where

\[
F = g_i \left\{ \beta_t \frac{\mu_t}{\sigma_t} \frac{\partial T}{\partial x_i} + \beta_t \frac{\mu_t}{\sigma_{ct}} \frac{\partial C}{\partial x_i} \right\}.
\]

(7)
Standard $k$-$\varepsilon$ turbulence model

The $k$-$\varepsilon$ transport equation is created from Navier-Stokes-equation on condition that the turbulence effect dominates over the whole flow field. The $k$-$\varepsilon$ turbulence model ensures the option to operate turbulence effects as transport equations.

The continuity equation for incompressible and source-free medium:

$$\frac{\partial u_i}{\partial x_i} = 0 , \quad (8)$$

where $u_i$ are velocity components, $x_i$ are coordinates, $i = 1, 2, 3$.

The conservation of momentum equations use Newton’s movement laws. The resultant of external forces, affecting the elementary volume, equals to the resultant of total momentum’s growth and total outgoing impulse from the elementary cell with reference to same elementary volume. These external forces on one hand are external stresses on the surface of the primary cell, on the other hand split force effects, like the force effect resulting from gravity:

$$\rho \frac{\partial u_i}{\partial t} + \rho \frac{\partial}{\partial x_j} (u_i u_j) + \frac{\partial p}{\partial x_i} \frac{\partial \tau_{ij}}{\partial x_j} - \rho F_i = 0 , \quad (9)$$

where $\tau$ is symmetrical liquid viscosity stress tensor, $\rho \cdot F_i$ is split force effect (e.g. gravity), for our purposes it is considered to be zero.

Liquid viscosity stress tensor in Newton’s medium:

$$\tau_{ij} = \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) , \quad (10)$$

where $\mu$ is the dynamic viscosity, Ns/m$^2$.

Equations (9), (10), and (11) describe Newton’s medium flow in laminar and turbulent case. If the computations were based on these equations, the model would have such a fine resolution for the investigation of smaller and greater fluctuations that in the end the necessary calculation power would be greater what an average computer could handle. Because of this, the Navier-Stokes equation’s time average modification has to be used. However, it can only be used for the calculation of large-scale fluctuations. Small fluctuations have to be described with the help of imminent or empirical methods. In 1883, Reynolds proposed and introduced the $f(x,t)$ value, which could manage the fluctuation’s size with an average in time.

Reynold’s filter can be stated in a more general form, where $f(x,t)$’s first component is the large-scale fluctuation’s average in the time, $\bar{f}(x,t)$, while the other component is the small-scale fluctuation’s average in time $f'(x,t)$:

$$f(x,t) = \bar{f}(x,t) + f'(x,t) . \quad (11)$$

This average-creating method can be understood as filter permeable at the bottom, which, in function of time, filters small-scale fluctuations.
The modified *Navier-Stokes* equation system and the continuity equation is as follows:

\[
\frac{\partial \mathbf{u}_i}{\partial x_i} = 0 \quad \text{and} \quad \frac{\partial \mathbf{u}_i}{\partial t} + \frac{\partial (\mathbf{u}_i \mathbf{u}_j)}{\partial x_j} + \frac{\partial}{\partial x_j} \left( R_{ij} - \frac{1}{\rho} \tau_{ij} \right) + \frac{1}{\rho} \frac{\partial p}{\partial x_i} = 0 ,
\]

where

\[
R_{ij} = \langle u'_i u'_j \rangle, \quad u'_i = u_i - \bar{u}_i, \quad p' = p - \bar{p}, \quad i, j = 1, 2, 3
\]

By introducing the concept of turbulent viscosity, which connects *Reynolds* stress and the gradient of the spatial mean velocity, and, following the suggestion of *Boussinesq* from 1887, the following can be stated:

\[
-R_{ij} = \nu_t \left( \frac{\partial u_i}{\partial x_j} - \frac{2 \cdot k}{3} \cdot \delta_{ij} \right),
\]

where \( \nu_t \) is the turbulent viscosity in \( \text{m}^2/\text{s} \).

Turbulent medium kinetic energy:

\[
k = \frac{1}{2} \sum R_{ii} = \frac{1}{2} \langle u'_i u'_i + u'_2 u'_2 + u'_3 u'_3 \rangle .
\]

Using these terms, the definition of the \( R_{ij} \) value is simplified to the calculation of the turbulent viscosity. However, turbulent viscosity depends on flow and not on the medium. The turbulent viscosity after the dimension analysis is:

\[
\nu_t = \frac{\mu_t}{\rho} = C_{\mu} \frac{k^2}{\varepsilon} .
\]

With the *k-\varepsilon* turbulence model it becomes possible to manage turbulent effects as a transport equation. It is an important advantage that numerical methods can handle transport equations and thus, besides the known transport (diffusion) processes, turbulence can be modelled as well.

Yet, the *k-\varepsilon* turbulence model does not provide satisfactory accuracy in the case of flows in the wall region. Therefore, the application of wall law cannot be avoided, adding more equations to the equation system.

**RESULTS**

Figure 4 and 5 show the temperature and velocity distributions and the temperature flow lines in the room.

It can be seen on Figure 4 that the air coming through the air inlet is heated up quickly and the temperature in the occupied zone is between 20 and 22 °C. Figure 5 shows that in the occupied zone velocities are far below 0.1 m/s.
Figure 4. Temperature distribution in the room

Figure 5. Velocity distribution in the room

The numerical data from the CFD simulation are collected in Table 1.

Table 1. Results of the modelling for winter case

<table>
<thead>
<tr>
<th></th>
<th>12</th>
<th>24</th>
<th>28</th>
<th>36</th>
</tr>
</thead>
<tbody>
<tr>
<td>Nominal heat output of the appliance, kW</td>
<td>12</td>
<td>24</td>
<td>28</td>
<td>36</td>
</tr>
<tr>
<td>Mass flow of the flue gas, kg/h</td>
<td>22.3</td>
<td>49.3</td>
<td>75.6</td>
<td>108</td>
</tr>
<tr>
<td>Temperature of the flue gas leaving the appliance, °C</td>
<td>101</td>
<td>121</td>
<td>122</td>
<td>118</td>
</tr>
<tr>
<td>Temperature of the flue gas in the beginning of the chimney’s indoor section, °C</td>
<td>88</td>
<td>104</td>
<td>112</td>
<td>108</td>
</tr>
<tr>
<td>Temperature of the flue gas in the beginning of the chimney’s outdoor section, °C</td>
<td>70</td>
<td>75</td>
<td>89</td>
<td>82</td>
</tr>
<tr>
<td>Temperature of the flue gas at the outlet point, °C</td>
<td>60</td>
<td>62</td>
<td>84</td>
<td>71</td>
</tr>
<tr>
<td>Air flow rate, m³/h</td>
<td>23</td>
<td>45</td>
<td>53</td>
<td>73.3</td>
</tr>
<tr>
<td>Pressure difference by the air-inlet, Pa</td>
<td>3.9</td>
<td>4.4</td>
<td>4.5</td>
<td>4.5</td>
</tr>
<tr>
<td>Average air temperature of the room, °C</td>
<td>24.1</td>
<td>23.6</td>
<td>23.5</td>
<td>23.6</td>
</tr>
<tr>
<td>Average air velocity in the room, m/s</td>
<td>&lt;0.1</td>
<td>&lt;0.1</td>
<td>&lt;0.1</td>
<td>&lt;0.1</td>
</tr>
</tbody>
</table>
DISCUSSION

Based on the data introduced in the table and on the distributions indicated in the figures the following can be stated:

- the average air temperature in the occupied zone is adequate (24 °C) and uniform,
- the relative air velocity in the occupied zone is far below 0.1 m/s,
- the air velocity in the occupied zone has a uniform distribution,
- 30 cm from the air-inlet the entering fresh air has a temperature that hardly differs from the average air temperature of the room,
- it is reasonable to place a heat-transfer appliance (accurately sized radiator) under the air-inlet so that the cool zone of the air-inlet can be reduced.

The investigation about appliances’ air supply and the publication of the design aiding results is of importance as in Hungary numerous B11 type appliances with atmospheric burner and draught hood, connected to a chimney with natural draught operate.

REFERENCES

6. CEN/TR 1749 European scheme for the classification of gas appliances according to the method of evacuation of the products of combustion (Types) Technical report, December 2005