Proceedings of the 4th WSEAS Int. Conf. on HEAT TRANSFER, THERMAL ENGINEERING and ENVIRONMENT, Elounda, Greece, August 21-23, 2006 (pp16-21)

Numerical modelling of air-supply and chimney of B_{11} type gas appliances

DR. LAJOS BARNA DR. LÁSZLÓ GARBAI RÓBERT GODA
Department of Building Service Engineering
Budapest University of Technology and Economics
1111 Budapest, Műegyetem rkp. 9.
HUNGARY

Abstract: In Hungary, most operating gas appliances are B_{11} type devices. In the case of these appliances, the coordinated operation of the gas appliance and the chimney, as well as the supply of combustion air raise important questions. Chimneys with natural draught are very sensitive to the changes in the amount of combustion air which may be caused by inside or outside ambient phenomena or by forced effects.

The presented model seeks to examine the section between the air inlet of the room and the outlet of the chimney (air inlet – room – gas appliance – chimney). For the purposes of modelling, a numerical simulation (CFD method) can be used. The aim of numeric modelling is to investigate velocity and temperature conditions around the flue gas outlet and in the room and, subsequently, to define design approaches and the requirements for different conditions.

Key words: air supply, chimney, gas appliances, CFD method, numerical modelling

1 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 as follows [8]:

– Appliances with atmospheric burner and draught hood, connected to a chimney with natural draught (e.g. B_{11}, B_{41}),
– Appliances, which have burners installed with ventilators, connected directly to the chimney, without draught hood (e.g. B_{23}, B_{33}, B_{53}).

In Hungary, the use of gas is of a significant proportion as compared to other energy sources. Small consumers – with a gas meter no greater than 20 m^3/h nominal volume flow, including domestic consumers – make up almost 50% of the market.

9-10 million gas appliances are estimated to operate in Hungary, most of which are connected to a chimney and have open combustion chamber (B_{11} type, Fig.1). 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. In recent years, this phenomenon has caused several accidents in Hungary, some of which turned out to be fatal.

Regulations have not been updated to follow the innovations of gas appliance designs and they do not include the drastic decrease of air-change rates due to air-tight windows and doors. This is why special attention is paid to the modelling of B_{11} type gas appliances. With a theoretically established background, the placement, design and operation of the appliance becomes easier.

The mathematical modelling and its' results for the B_{11} type gas appliances are included in last year's WSEAS Conference Proceedings ([4], [5]); in this paper, the theoretical basis of numerical modelling is summarized.

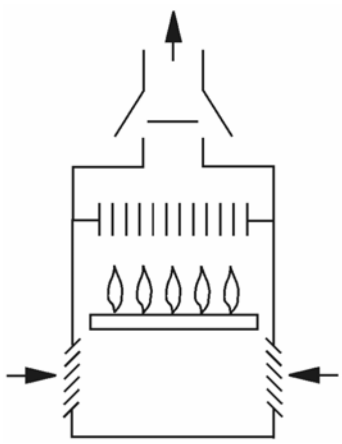

Fig.1 B_{11} type gas appliance according to [8]

2 Problem Formulation

The presented model seeks to examine the section between the air inlet of the room and the outlet of the exhaust (air inlet – room – gas appliance – chimney).

Chimneys with natural draught are very sensitive to the changes in the amount of combustion air which may be caused by inside or outside ambient phenomena or by forced effects. In extreme cases, even the minimum amount of air required for the burning process is unavailable, which means that the appliance will not work at the adequate operating point. 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 changes in the inside or outside ambient conditions, numerical investigation can be used. For the numerical investigation, we use the „air as fluid" method. (CFD, Computational Fluid Dynamics). With the help of CFD, the phenomena can be studied in what is virtually a computational environment.

The aim of numeric modelling is to examine the velocity and temperature conditions in the room, in the chimney and around the flue gas outlet. The results of the calculations can help in defining designing approaches and the requirements for different conditions.

3 Problem Solution

Steps of modelling:

- creating the geometry of the model,
- stating the differential equations for the numeric model,
- developing the CFD model,
- creating the CFD model,
- modelling the air supply and flue gas removal of a gas appliance for different conditions and operation modes.

3.1 The geometry of the model

For the modelling of the B_{11} type gas appliance a conventional sized room is used, in which the appliance is the only equipment (Fig.2). 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.

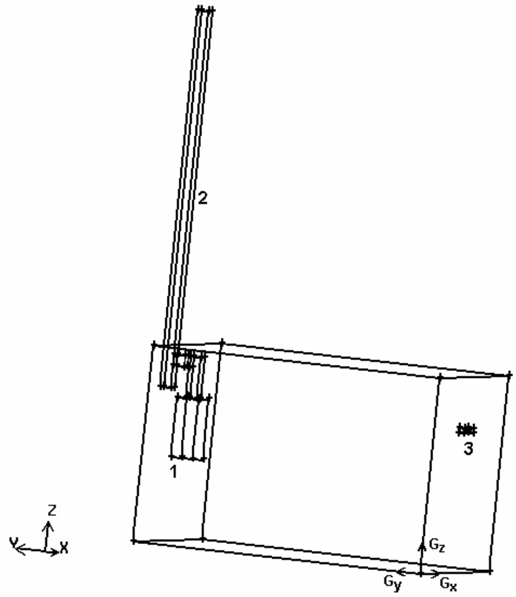

Fig.2 The geometric model used for the examination of B_{11} type gas appliances
1 – wall-mounted gas appliances, 2 – chimney, 3 – air inlet

3.2 Numeric model

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

In the numeric model, the walls of the room are adiabatic, that is, no mass or heat transfer takes place through them. Combustion air enters the indoor environment via a porous volume, which has a certain preset resistance characteristic. The gas appliance itself is also represented by a special volume, in which heat is discharged evenly and constantly, according to the performance of the appliance. The volumetric heat source (combustion) and the flow induced by the chimney leaves the room through the chimney. The chimney is a volume element, which has resistance and heat conductive properties.

3.2.1 Differential equations for the numeric model

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-ε) 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 formulated as follows:

Continuity:

$$\mathrm{div}\,(\rho \cdot u_i) = 0 \tag{1}$$

where:
ρ – air density.
u_i – air velocity components in *x, y, z* direction

Equation of movement:

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

where
μ – viscosity,
μ_t – turbulent viscosity,
p – pressure,
k – kinetic energy,
δ_{ij} – *Kronecker* symbol.

Equation of energy:

$$\frac{\partial}{\partial x_i}(\rho \cdot u_i \cdot h) = \frac{\partial}{\partial x_i}\left\{\left(\frac{\mu}{\sigma} + \frac{\mu_t}{\sigma_t}\right)\frac{\partial h}{\partial x_i}\right\} + Q \tag{3}$$

where
h – enthalpy,
Q – quantity of heat per volume,
σ_t – factor, depends on *Prandtl*- and *Schmidt*-numbers.

Concentration of pollution:

$$\frac{\partial}{\partial x_i}(\rho \cdot u_i \cdot C) = \frac{\partial}{\partial x_i}\left\{\left(\frac{\mu}{\sigma_{cl}} + \frac{\mu_t}{\sigma_{ct}}\right)\frac{\partial C}{\partial x_i}\right\} + C_L \cdot \rho \tag{4}$$

where
σ_{cl} – factor, depends on *Prandtl*- and *Schmidt*-numbers in laminar flow,
σ_{ct} – factor, depends on *Prandtl*- and *Schmidt*-numbers in turbulent flow,
C – medium concentration of pollution in the air.

Turbulent viscosity:

$$\mu_t = K \cdot \rho \cdot \frac{k^2}{\varepsilon} \tag{5}$$

where
K – constant
ε – dissipation of kinetic energy.

Turbulent kinetic energy:

$$\frac{\partial}{\partial x_i}(\rho \cdot u_i \cdot k) = \frac{\partial}{\partial x_i}\left\{\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_i}\right\} - \\ - K_4 \cdot \rho \cdot \varepsilon + \mu_t \frac{\partial u_i}{\partial x_j}\left\{\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right\} + F \tag{6}$$

where
σ_k – kinetic energy factor.

Dissipation of turbulent kinetic energy:

$$\frac{\partial}{\partial x_i}(\rho \cdot u_i \cdot \varepsilon) = \frac{\partial}{\partial x_i}\left\{\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_i}\right\} - K_2 \cdot \rho \cdot \frac{\varepsilon^2}{k} + \\ + K_1 \cdot \mu_t \frac{\partial u_i}{\partial x_j}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\frac{\varepsilon}{k} + K_3 \cdot F \cdot \frac{\varepsilon}{k} \tag{7}$$

where

$$F = g_i\left\{\beta \frac{\mu_t}{\sigma_t}\frac{\partial T}{\partial x_i} + \beta_c \frac{\mu_t}{\sigma_{ct}}\frac{\partial C}{\partial x_i}\right\}. \tag{8}$$

3.2.2 Standard *k-ε* turbulence model

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

The continuity equation for incompressible and source-free medium:

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{9}$$

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

The conservation of momentum equations use the *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 are, on the one hand, external stresses on the surface of the primary cell and, on the other, split force effects, like the force effect resulting from gravity:

$$\rho \cdot \frac{\partial u_i}{\partial t} + \rho \cdot \frac{\partial}{\partial x_j}(u_i u_j) + \frac{\partial p}{\partial x_i} - \frac{\partial \tau_{ij}}{\partial x_j} - \rho \cdot F_i = 0 \tag{10}$$

where
τ – symmetrical liquid viscosity stress tensor,
ρF_i – split force effect (e.g. gravity), for our purposes it is considered to be zero.

Liquid viscosity stress tensor in *Newton*'s medium:

$$\tau_{ij} = \mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \tag{11}$$

where
μ – 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, necessary calculation power would be greater than 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.

$$f(x,t) = \bar{f}(x) + f'(x,t) \tag{12}$$

$$\bar{f}(x) = \langle f \rangle = \frac{1}{T} \int_{-T/2}^{T/2} f(x,\tau) d\tau \tag{13}$$

$$\langle f' \rangle = 0 \tag{14}$$

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 $f'(x,t)$.

$$f(x,t) = \bar{f}(x,t) + f'(x,t) \tag{15}$$

$$\bar{f}(x,t) = \langle f \rangle \tag{16}$$

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:

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0 \text{ and} \tag{17}$$

$$\frac{\partial \bar{u}_i}{\partial t} + \frac{\partial (\bar{u}_i \bar{u}_j)}{\partial x_j} + \frac{\partial}{\partial x_j}(R_{ij} - \frac{1}{\rho}\bar{\tau}_{ij}) + \frac{1}{\rho}\frac{\partial \bar{p}}{\partial x_i} = 0 \tag{18}$$

where

$$R_{ij} = \langle u_i' \, u_j' \rangle$$
$$u_i' = u_i - \bar{u}_i$$
$$p' = p - \bar{p}$$
$$i, j = 1, 2, 3$$

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:

$$-R_{ij} = \nu_t \left(\frac{\partial \bar{u}_i}{\partial x_j} \frac{\partial \bar{u}_j}{\partial x_i} \right) - \frac{2}{3} k \cdot \delta_{ij} \tag{19}$$

where
ν_t – turbulent viscosity, m^2/s.

Turbulent medium kinetic energy:

$$k = \frac{1}{2} \sum R_{ii} = \frac{1}{2} \langle u_1' u_1' + u_2' u_2' + u_3' u_3' \rangle . \tag{20}$$

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:

$$\nu_t = \frac{\mu_t}{\rho} = C_\mu \frac{k^2}{\varepsilon} , \tag{21}$$

Dissipation of medium turbulent viscosity:

$$\varepsilon = \nu \left\langle \frac{\partial u_i'}{\partial x_i'} \frac{\partial u_i'}{\partial x_j'} \right\rangle . \tag{22}$$

The two turbulent characteristics, k and ε satisfy the following transport equation in every point of the flow space:

$$\frac{\partial k}{\partial t} + \bar{u}_i \frac{\partial k}{\partial x_i} - \frac{\partial}{\partial x_i}\left(\frac{\nu_t}{\sigma_k} \frac{\partial k}{\partial x_i} \right) - \nu_t S + \varepsilon = 0 \tag{23}$$

$$\frac{\partial \varepsilon}{\partial t} + \bar{u}_i \frac{\partial \varepsilon}{\partial x_i} - \frac{\partial}{\partial x_i}\left(\frac{\nu_t}{\sigma_\varepsilon} \frac{\partial k}{\partial x_i} \right) - C_{\varepsilon 1} \nu_t \frac{\varepsilon}{k} S + C_{\varepsilon 2} \frac{\varepsilon^2}{k} = 0 \tag{24}$$

and

$$S = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} \frac{\partial u_j}{\partial x_i} \right)^2 , \text{ i, j = 1, 2, 3.} \tag{25}$$

The standard values of the model's empirical constants are:
$C_\mu = 0{,}09$ $C_{\varepsilon 1} = 1{,}44$ $C_{\varepsilon 2} = 1{,}92$
$\sigma_k = 1{,}0$ $\sigma_\varepsilon = 1{,}3$

The *k-ε transport equation* was obtained based on the *Navier-Stokes* equations, with the supposition that turbulent effects are dominant in the whole flow area.

With the *k-ε 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-ε 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.

3.3 The development of a CFD numeric simulation model

Similar to the mathematical-physical description of the problem, geometric data provide the basis for the creation of the model. A 3D net can be elaborated to suit the model (Fig.3). The net resolution follows geometry, the boundary, and the initial conditions. At these specific points, at which high gradients can be expected, the mesh should be refined, thereby decreasing the instability of the numerical calculation and the required calculation time.

During simulation, as the flows and gradients become increasingly accurate, the initial mesh must be refined according to the demands that will have to be verified by the following steps of calculation.

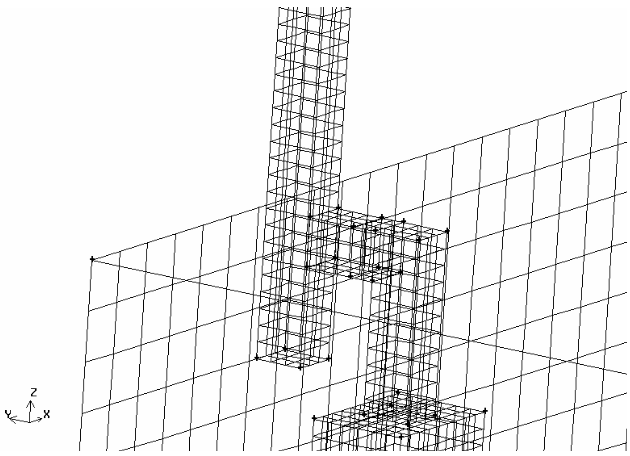

Fig.3 The mesh model

4 Calculation results

Calculations have been carried out using the model in a room with a B_{11} type boiler and in the chimney attached to the device in order to determine the evolving flow and temperature conditions.

The nominal heat output of the examined wall-mounted boiler is:

Q_N = 12, 18, 24, 30, 40 kW

The main data of the geometric model are:

– Volume of the room: 15 m^3
– Total height of chimney above the connection: 6 m, of which 4 m are in the building and 2 m continue outside the roof.

In the course of the calculations carried out so far, ambient air temperature varied between +32 °C and –15 °C.

The calculations give results of the changes in the following parameters:

– magnitude and direction of air velocity in the room, between the air inlet and the device,
– flow velocity of the flue gas in the connecting flue pipe and the chimney,
– flue gas temperature in the connecting flue pipe and the chimney, between the connection and the outlet.

Figures 4–6 illustrate some of the calculation results.

Fig. 4 illustrates flue gas temperature calculated in the lower connecting flue pipe and chimney section (ambient air temperature: +32 °C). Flue gas temperature in the lower section that runs within the building is considerably greater than flue gas dew point temperature.

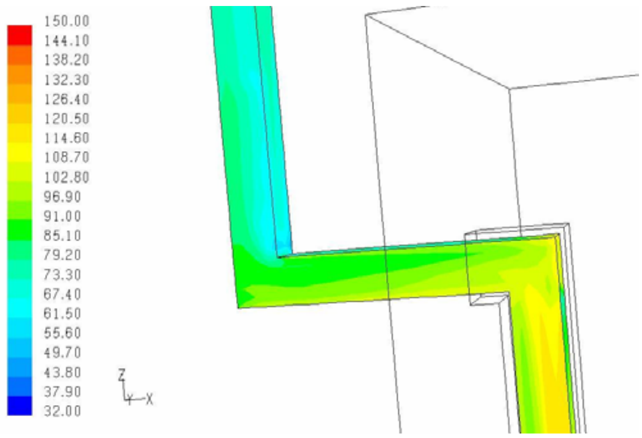

Fig.4 Wall temperature in the connecting flue pipe and at the bottom of the chimney

Fig. 5 illustrates velocity vectors calculated in the lower connecting flue pipe and chimney section. Flue gas velocity is about 0,8 m/s.

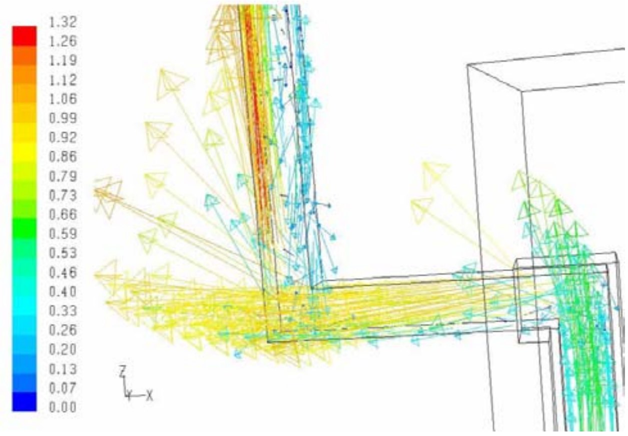

Fig. 5 Flue gas velocity in the connecting flue pipe and at the bottom of the chimney

Fig. 6 indicates flue gas temperature calculated on the basis of the winter sizing outside temperature of –15 °C at the point where the chimney leaves the building (on the left) and at the chimney outlet (on the right). With normal chimney structure, flue gas temperature is so low amongst adverse weather conditions that condensation occurs within the chimney. Therefore, additional insulation or a flue duct insusceptible to condensation is required.

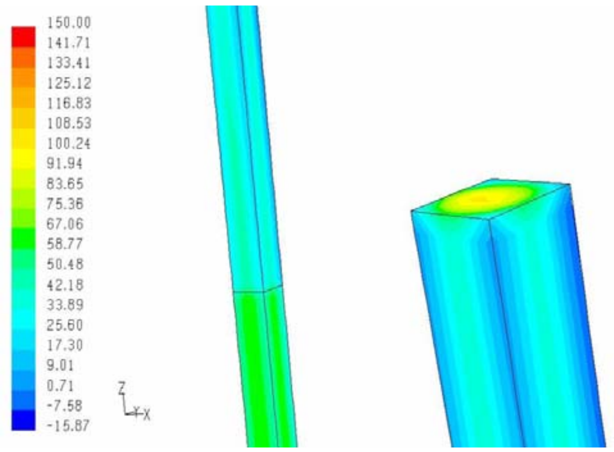

Fig. 6 Flue gas temperature in the flue duct at the point of leaving the building and the chimney outlet

5 Conclusion

The present paper introduced the elaboration of a CFD numeric simulation model that can be used for the examination of air supply and flue gas removal for the most widely used gas appliances in Hungary, that is, B_{11} type devices with draught hoods and chimney connection. Numeric modelling, in order to help the design process (i.e. placement of the appliance in the room, etc.), seeks to develop design conditions and the requirements.

References:

[1] Garbai, L. – Barna, L. – Vigh, G., Modelling of Instacioner Conditions in a Space Heated by an Individual Gas Boiler, *Gépészet 2004 Conference,* Budapest University of Technology and Economics, 2004, Proceedings pp. 264-268.

[2] Garbai, L. – Barna, L., Air Supply Modelling in a Room Heated by an Individual Gas Boiler, *Energy for Buildings 6th International Conference,* Vilnius, Lithuania, 2004, Proceedings pp. 428-435.

[3] Garbai, L. – Barna, L. – Varga B., Atmoszférikus gázkazánok égéstermék-elvezetésének és levegőellátásának méretezése (Sizing of the Chimney and the Air Supply for Atmospheric Gas Appliances) in: *Magyar Épületgépészet*, Vol. LIII 2004/11, p. 3-7.

[4] Garbai, L. – Barna, L., Modelling of non steady state conditions in a room heated by a gas boiler. *3rd IASME /WSEAS Int. Conf. on Heat Transfer, Thermal Engineering and Environment*, Corfu Island, Greece, August 20-22, 2005

[5] Garbai, L. – Barna, L., Operation of gas boilers at non-steady-state conditions. *IASME Transactions* Issue 9, Volume 2, November 2005, ISSN 1790-031X p. 1801-1809

[6] Barna, L. – Goda, L., B_{11} csoportba tartozó gázfogyasztó készülékek levegőellátásának és égéstermék-elvezetésének numerikus modellezése (Numerical modelling of air-supply and flue gas evacuation of B11 type gas appliances) in: *Magyar Épületgépészet*, Vol. LV 2006/5, p.9-12

[7] Garbai, L. – Barna L., Modelling of Air Supply Conditions of Gas Boilers With Opened Combustion Chambers, *INFUB 7th European Conference*, Porto, Portugal, 18-21 April 2006, Poster Presentation

[8] 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