18th International Research/Expert Conference
”Trends in the Development of Machinery and Associated Technology”
TMT 2014, Budapest, Hungary 10-12 September, 2014

CHIMNEYS WITH REDUCED DRAFT AND VAPOR CONDENSATION OF WET FLUE GAS

Nijaz Delalić, Muris Torlak, Ejub Džaferovic

**Faculty of Mechanical Engineering, Sarajevo
Vilsonovo setaliste 9, BA-71000 Sarajevo**

ABSTRACT
In order to increase energy efficiency of heating plants and obtain lower CO_2 emission into atmosphere, there is a tendency to keep exhaust-gas temperature lower than its typical value referred to as "acid dew point". As a result, vapor condensation of wet flue gas appears in chimneys. Condensation occurs when the surface temperature is below the dew point of the vapor-gas mixture. Therefore, Vapor-Liquid Equilibrium models are required in order to determine the dew point of the mixture. A numerical model of turbulent vapor-air mixture flow through a chimney is presented. The velocity, temperature, as well as species fields are calculated via a finite-volume CFD code, whose validation is conducted employing a real object in chimney. The experiment on heat and mass transfer of a two-phase, two-component system is done Comparisons of gas temperature and pressure at the wall are shown, revealing a good prediction obtained by the model presented.
Keywords: energy efficiency, chimney, vapor condensation

1. INTRODUCTION
Increase of the emission of CO_2, which is mostly the result of the combusted fossil fuels into the atmosphere, exponentially increases. This trend of increase of CO_2 presents immense danger for the planet. For this reason the organized part of the world must resist this trend of the increase of emission. Some measures to stop increase of the emission of CO_2 must be undertaken, and then measures to reduce emission of CO_2 into the atmosphere should follow. One of the very important ways to decrease the emission of CO_2 is to increase energy efficiency.
The development of new techniques for heating buildings systems, like high performance generator, burners with modulated functioning, allow design henceforth installation with high thermal efficiency. Such installations require all energy recuperation and consequently lower temperatures of exhaust flue gases. This causes vapor condensation of flue gases. Use of latent heat of the vapor condensation from exhaust gases as well as the measure for increasing the energy efficiency is very suitable regarding the combustion of the gaseous fuels. But, problem is very complex, because as it involves more coupled phenomena: two-phase flow, multi component flow, turbulence, condensation, liquid film, diffusion vapor of water from centerline to the wall of the chimney conduit. There are not many papers in this topic. Researchers, Junker [2] and Pitschak [3] provided numerical and experimental work for prediction thermal field flows, the heat transfer of the flue gases and diffusion vapor across the wall of the chimney. Maref [4] employed numerical method of finite volume for prediction of thermal field flows, the heat transfer of the flue gases in the simple chimney. The aim of this work is experimental and numerical prediction of the thermal field flows, the heat and mass transfer of the flue gases to the wall. Wet flue gas was simulated by vapor-air mixture. These cases were studied.

2. EXPERIMENTAL STUDY
A schema of the experimental apparatus is shown in Fig. 2. Heat and mass transfer experimental apparatus has been developed on the basis of the apparatus which is similar as described by Delalic [1]. The system considers air–vapor flow. Compressed air passes through an air filters and later

through an Inlet unit. Secondary air is used for the cooling test section, while the primary air is used like flue gas in the test section. The electrical power is dissipated in the Flow development section, which acts as a resistance element and ensures wall heat flux for heating test-gas.

Measurement of Gas parameters

The outer wall temperature distribution of the test section is measured by twenty-nine chromel- alumel thermocouples of 24 gauge (0.51 mm dia) which are in contact with the outer surface (see Fig. 1). The thermocouples were connected to the output of data acquisition system, by InstruNET, PC-controller, model I-200, and network device, model I-100. Thermocouples are placed at the centerline at the inlet and the outlet of the test section to estimate the bulk fluid temperature. At the inlet the bulk fluid temperature is taken to be equal to the measured centerline temperature.

Flow meters for gas and cooling air have a range from 0.5 to 5.5 m^3/h and accuracy of ±2%. A pressure measurement is made at the outlet of the flow meters with U-tube manometer, and the pressure drop across the test section is measured by micro manometer with inclined pipe. Ambient pressure and temperature measurement were necessary and they are made in the loop. The humidity probes are placed at the inlet and the outlet of the test section to measured humidity of gas. The humidity meter used in the experiment has measurement range of 10 to 90% relative humidity with accuracy ±2%. The condensing rate was calculated from the deference between inlet and outlet humidity.

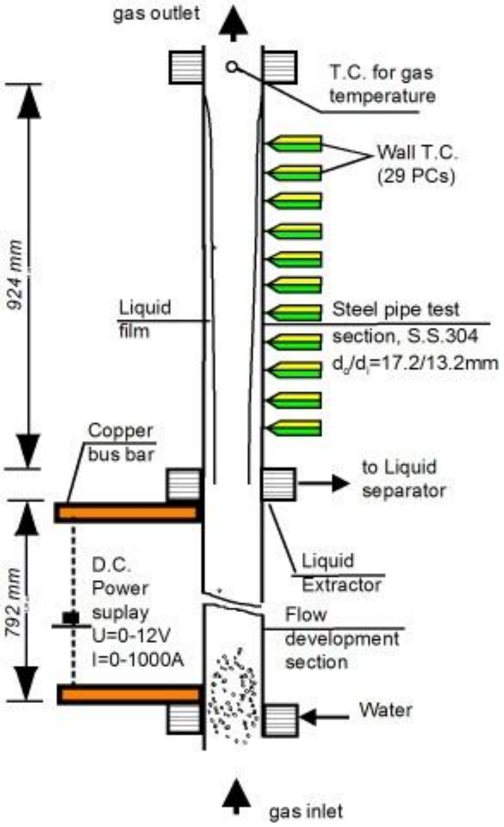

Figure 1. Experimental set-up for the heat and mass transfer studies into flue conduit

The following cases were studied in the experiment (see Fig.2); I - upward gas flow without condensation, II - upward gas flow and counter air flow without condensation, III- upward gas flow and counter air flow with condensation

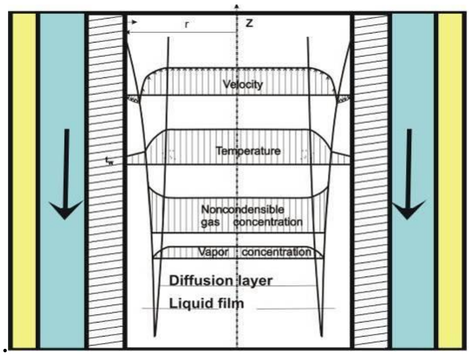

Figure 2. Physical model of experimental cases

3. EXPERIMENTAL CONDITIONS AND RESULTS

Table1. Test conditions for the heat and mass transfer in flue conduit

	Case I	Case II	Case III
Gas flow rate (m_n^3/h)	1.94-5.17	1.94-5.17	1.94-5.17
Cooling Air flow rate (m_n^3/h)	0	1.755-4.68	1.755-4.68
Gas inlet temperature (°C)	80	80	60
Air inlet temperature (°C)	0	20	20
Gas outlet temperature (°C)	64.5-72.3	56.0-60.5	56.3
Air outlet temperature (°C)	0	44.6-45.3	
Re- number	2937-6089	2937-6089	2937-6089
Vapor volume fraction, %	1	1	17.3

In the case I difference of inlet-outlet gas temperature is 8-15 K. The negative slope in the measured T_w near the inlet of the test section is due to longitudinal heat loss. In the case II difference of inlet-outlet temperature is higher than case I and is fixed at 20-24 K.

The wet flue gas was simulated by vapor-air mixture produced from inlet unit. The excess air coefficient for combustion is 1.1 and the water vapor portion in the gas is 17.3% in the experiments.

When the temperature of the flue gas is lower or the gas approaches saturation, some water vapor will be condensed. Because the partial pressure of the water vapor in the wet flue gas is small, the resistance to the mass transfer is large and condensation of the vapor is difficult. In the experiments conducted, about 20 to 30% water vapor was condensed and total heat transfer was about doubled.

The heat transfer was influenced by the water concentration in gas mixture. In the experimental investigation, when cold air was used to cool wet flue gas, the contribution of condensation heat transfer was about 2 to 2.5 times more than convection heat transfer. Fig.3. shows comparison between distributions of wall and flue gas temperature in the case II and III. Fig. 4. shows pressure drop in case without condensation and with condensation. Pressure drop is higher in the case with condensation than case without condensation.

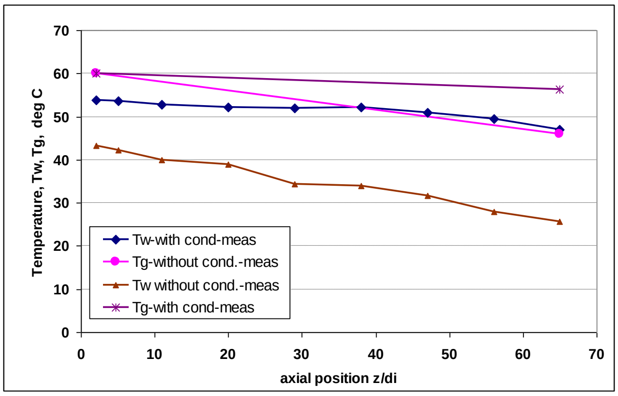

Figure 3 Comparison experimental wall and gas temperature for case II and case III

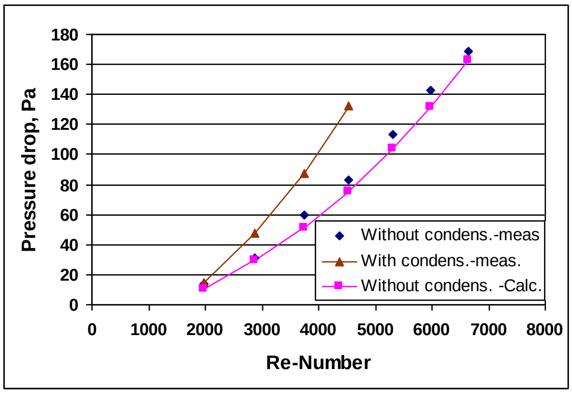

Figure 4. Pressure drop in the conduit with and without condensation

4. NUMERICAL STUDY

Formulation of the problem

The aim of the numerical model is to predict the gas flow, thermal field and wall temperature in the conduit. The physical contributions of conduction in the wall and mixed convection inside the conduit are described by these equations. The following assumptions are adopted; smoke is assumed to be a part of dry air, the typical Re-number at the entrance is from 2937 to 6776, the flow is fully turbulent, incompressible and 2D axisymmetric, there is no heat source in the energy equation, and the thermo-physical fluid properties are constant.

The equations for the formulation of the problem are written in cylindrical polar coordinates (r,Θ,z).

Conservation of mass:

$$\frac{\partial}{\partial z}(W)+\frac{1}{r}\frac{\partial}{\partial r}(rV)=0$$

Conservation of Axial-directional Momentum

$$\rho\left(\frac{\partial W}{\partial t}+V\frac{\partial W}{\partial r}+W\frac{\partial W}{\partial z}\right)=-\frac{\partial P}{\partial z}+\frac{1}{r}\frac{\partial}{\partial r}\left[r\mu_{eff}\left(\frac{\partial W}{\partial r}+\frac{\partial V}{\partial z}\right)\right]$$

Conservation of Radial-directional Momentum

$$\rho\left(\frac{\partial V}{\partial t}+V\frac{\partial V}{\partial r}+W\frac{\partial V}{\partial z}\right)=-\frac{\partial P}{\partial z}+\frac{1}{r}\frac{\partial}{\partial r}\left[2r\mu_{eff}\frac{\partial V}{\partial r}\right]-2\mu_{eff}\frac{V}{r^2}$$

Conservation of energy

$$\frac{\partial T}{\partial t}+V\frac{\partial T}{\partial r}+W\frac{\partial T}{\partial z}=\frac{1}{r}\frac{\partial}{\partial r}\left[\left(a+\frac{\nu_t}{\mathrm{Pr}_t}\right)r\frac{\partial T}{\partial r}\right]$$

Where is U,V,W velocity in x, r and z directions, respectively, a thermal diffusivity, P-pressure, Pr_t-Prandtl number turbulent, t - time variable, T – temperature, ρ-density, μ_0, μ_t – dynamic and turbulent viscosity, respectively.

Numerical code

The momentum and temperature field were solved by using the commercial software COMET based on the well-known finite-volume method described by Patankar [6], Demirdzic, Muzaferija and Peric [7,8], using the $k-e$ turbulence model.

Boundary conditions

The experimental data profiles for the mean stream wise velocity and temperature are introduced at the entrance section. At the exit section, the relative pressure is fixed to zero.

The external temperature is fixed to the constant ambient value T_{amb}=293 K. The temperature at the outer wall T_{wout} is computed by establishing the thermal balance. This corresponding thermal resistance at the outer wall is fixed at 0.125 m^2K/W.

Numerical results

Distributions of calculated wall temperature and gas temperature along the test section for Re=6776 are shown on Fig.5. Good agreement is found between experimental data and calculation. Prediction of outlet gas temperature is made in the dependence from Re-number, as shown on the Fig. 6. Calculated temperature is little higher, because axial heat loss at the test section. Although, experimental data are found 2-3 K below calculated values.

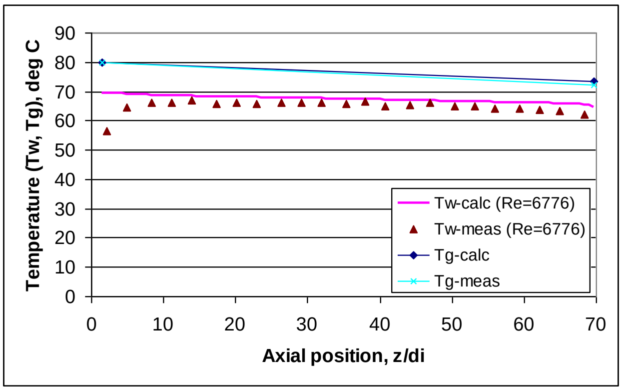

Figure 5. Experimental and numerical wall and gas temperature, Re=6776

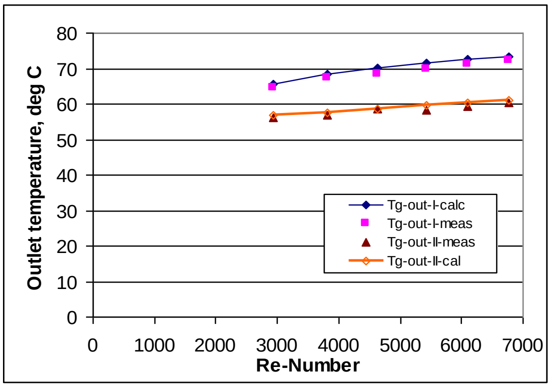

Figure 6. Experimental and numerical outlet gas temperature

5. CONCLUSIONS

The aim of this work is to provide experimental and numerical analysis of the thermal field flows, the heat and mass transfer of the flue gases to the wall. Wet flue gas was simulated by vapor-air mixture. Experiment is provided for three cases; I - upward gas flow without condensation, II - upward gas flow and counter air flow without condensation, III- upward gas flow and counter air flow with condensation. Numerical analysis is provided for first two cases. Comparison between experimental data and numerical calculation are giving good agreement. When water vapor concentration is high in the wet flue gas, the flue gas temperature drops and the thermal efficiency is raised. In the experimental range vapor concentration, the heat transfer coefficient is twice as great as the single-phase convection heat transfer. In that case the condensation heat transfer cannot be neglected. Pressure drop is higher in the case with condensation than in the case without condensation. The core of the future work will be aimed to numerical analysis of the condensation vapor in the wet flue gas.

6. REFERENCES

[1] N. Delalic: Experimental investigation of heat and mass transfer in two-phase, dispersed flow at low pressure and low mass velocities. M. Sc. Thesis, Univ. of Sarajevo, Bosnia and Herzegovina, 2002.

[2] B. Junker: Numerische Berechnung des gekoppelten Impuls-, Wärme- und Stofftransportes in Schornsteinen von Erdgas- und Erdgas-/Wasserstoff-Feuerungen. Dissertation, Technische Universität Hamburg-Harburg, 1996.

[3] B. Pitschak: Experimentelle und theoretische Untersuchungen zum instationären Wärme und Feuchtehaushalt von Schornsteinen. Dissertation, Ruhr-Universität Bochum, 1996.

[4] W. Maref; H. Cherkaoui, P. Crausse, H. Boisson: Building and Environment, Vol. 38 (2003), p. 763–770

[5] SV. Patankar: Numerical heat transfer and fluid flow, Washington USA, Hemisphere Publishing Company,

[6] I. Demirdzic and S. Muzaferija: Introduction to Computational Fluid Dynamics, Mechanical Engineering Faculty, University of Sarajevo, Bosnia and Herzegovina, 1997.

[7] Comet-CFD-Commercial software, Institute of Computational Continuum Mechanics GmbH, Hamburg, Germany 1998.