Asociación Argentina de Mecánica Computacional

Mecánica Computacional Vol XXXII, págs. 1071-1079 (artículo completo)
Carlos G. García Garino, Aníbal E. Mirasso, Mario A. Storti, Miguel E. Tornello (Eds.)
Mendoza, Argentina, 19-22 Noviembre 2013

COMPUTATIONAL FLUID DYNAMICS (CFD) ANALYSIS OF A FLOW STRAIGHTENER FOR A HEATER CHIMNEY

Federico Bacchi and Ana Scarabino

Grupo Fluidodinámica Computacional, Universidad Nacional de La Plata, Calle 116 e/47 y 48, 1900 La Plata, Argentina, gfc@ing.unlp.edu.ar, http://www.gfc.ing.unlp.edu.ar

Keywords: chimney, cyclonic flow, flow straightener.

Abstract. This work presents a numerical analysis by means of computational fluid dynamics (CFD) of the flow within and outside a chimney of low length/diameter ratio with a lateral inlet for combustion gases. A partially blocked gases entrance and cyclonic flow present in the chimney in its actual configuration prevent the system to fulfill particulate flow control regulations, in particular concerning the velocity vector inclination in the control plane where monitoring is carried out.

A flow straightener device is proposed which reduces, in the numerical simulations, the inclination of streamlines in the flow control plane from more than 35 degrees to less than 5 degrees, meeting the criteria stated in environmental regulations for particle emission control, without introducing excessive pressure losses in the flow. Results of the analysis include the flow configuration, velocity and pressure distributions and helicity distribution, the latter as a measure of the intensity of cyclonic flow.

In this problem, gas enters the chimney from a lateral chamber. Numerical simulation of the present situation highlighted three main problems:

- Cyclonic flow in the chimney, generating helycoidal streamlines where the velocity vector inclination is beyond acceptable limits,
- Flow acceleration in the sector opposite to entrance, due to the effect of the lateral inlet and the deviation that gas suffers after impinging the wall,
- Generation of a horizontal vortex in the pre-entrance chamber, due to obstructions in the inlet ducts.

From this analysis, possible solutions were proposed and studied numerically. The one presented in this work, a simple straightener put immediately after the chimney entrance, plus the cleaning of all obstructed ducts, show in the results a significant improvement in the flow quality and alignement. The computational domain included the inlet chamber, the chimney and the atmosphere in a cylindrical region of approximately 55 chimney diameters in width and 8 chimney heights in height.

The numerical simulation was performed with Ansys Research package. A pressure-based solver was used for compressible flow transient analysis, with second order discretization for space and time, and the “species transport” method was employed for computing the mixing of two species: combustion gas with known properties, and air. The variations with temperature of viscosity, thermal conductivity and specific heats for the gas were approximated with polynomial expressions interpolating known values. Gas density was computed from an equation of state. The k-epsilon turbulence model was used. The mesh was locally refined in order to achieve grid-independent results. The time step for convergence was 1e-5 s.

1 INTRODUCTION

International standards for control of pollutants and particulate flow into the atmosphere require continuous monitoring of gases in discharge chimneys (US EPA CFR 40, IRAM 29230). Among other requirements, the gas velocity must be measured in a plane at a specified distance from the discharge outlet. The flow in this plane must be as uniform as possible and the stream lines must be vertical, or inclined by no more than a few degrees. A common problem is the occurrence of cyclonic flow, i.e. flow organization in one or more longitudinal vortices. The strong rotational velocity component makes the streamlines helical, invalidating conventional speed measurements with pitot tubes or other methods which can be sensitive to sensor misalignment with the flow.

The availability of computational resources has led to an increasing number more numerical studies are performed to optimize the designs of all types of chimneys and ducts for fluid transport (Kazansky et al, 2003; Harris, 2007; Andreozzi, 2010).

This work in particular presents a numerical analysis of the flow within and outside a chimney of low length/diameter ratio with a lateral inlet for combustion gases. A partially blocked gases entrance and cyclonic flow present in the chimney in its actual configuration prevent the system to fulfill particulate flow control regulations. The flow is studied numerically in the system actual condition and in an ideal condition of free entrance, for different wind velocities. A flow straightener device is proposed which reduces, in the numerical simulations, the inclination of streamlines in the flow control plane from more than 35 degrees to less than 5 degrees, meeting the criteria stated in environmental regulations without introducing excessive pressure losses in the flow. Results of the analysis include the flow configuration, velocity and pressure distributions and helicity distribution, the latter as a measure of the intensity of cyclonic flow.

In this problem, gases that enter the chimney come from a lateral chamber. Numerical simulation of the present situation highlighted three main problems: cyclonic flow in the chimney, flow acceleration in the sector opposite to entrance, and the generation of a horizontal vortex in the pre-entrance chamber, due to obstructions in the inlet ducts.

From this analysis, possible solutions were proposed and studied numerically. The one presented in this work, a simple straightener put immediately after the chimney entrance, plus the cleaning of all obstructed ducts, show in the results a significant improvement in the flow quality and alignment.

2 METHODOLOGY

The computational domain included the inlet chamber, the chimney and the atmosphere in a cylindrical region of approximately 55 chimney diameters in width and 8 chimney heights in height (figure 1). This wide external domain allows more realistic results than those obtained by simply imposing a pressure condition at the chimney outlet, and it also allows to analyze the influence of wind in the chimney operation and the plume dispersion.

The assembly entrance chamber-chimney was discretized in a multiblock scheme that allows to modify sectors of the domain according to the different proposed modifications, without the need of a whole re-meshing, what allows saving CAD and meshing time.

The numerical simulation was performed with Ansys Research package. A pressure-based solver was used for compressible flow transient analysis, and the “species transport” method was employed for computing the mixing of two species: combustion gas with known properties, and air. The variations with temperature of viscosity, thermal conductivity and specific heats for the gas were approximated with polynomial expressions interpolating known values. Gas density was computed from an equation of state based on pressure and temperature, as a perfect gas. The k-epsilon turbulence model was used. The mesh was locally refined until it included 4e6 elements, in order to achieve grid-independent results. Figure 2 shows local mesh refinements. The time step for convergence was 1e-5 s.

Boundary conditions were walls, mass flow at the chamber inlet and pressure far field for the atmosphere. Wind velocity was initially set to zero, and in a further analysis, to a uniform velocity of 10 m/s from South West, as indicated in Fig. 3.

A reference plane was defined at a distance of two diameters to the chimney exhaust for the analysis of local velocity and other variables distribution, and eventual future comparisons with experimental measurements.

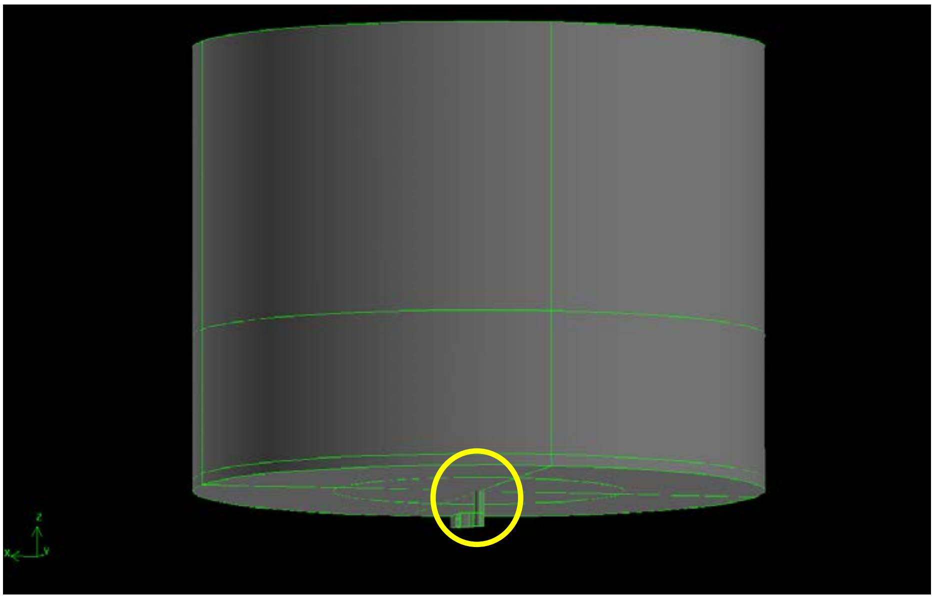

Figure 1: Computational domain. The chimney is in the yellow circle.

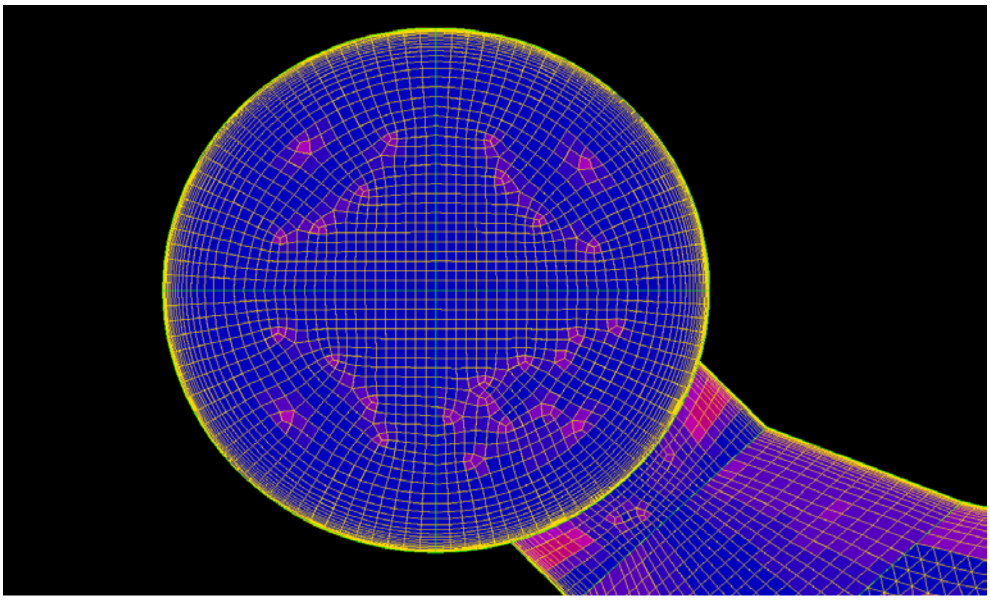

Figure 2: Detail of mesh refinement in the chimney.

The flow inlet to the chimney lateral pre-entrance chamber consists of a number of ducts. After a period of operation, if no maintenance is carried out, the ducts near the chimney become obstructed by deposits of residuals, leaves and dust carried by the wind when the chimney is not operating. The system still operates with acceptable efficiency, but the flow within the chimney is affected by this condition at the entrance.

In order to model the partial obstruction of the entrance ducts, the inlet was divided in two sections as shown in figure 3. The operating conditions for the chimney are then “free inlet” and “partially blocked inlet”. In both cases the total mass flow is the same.

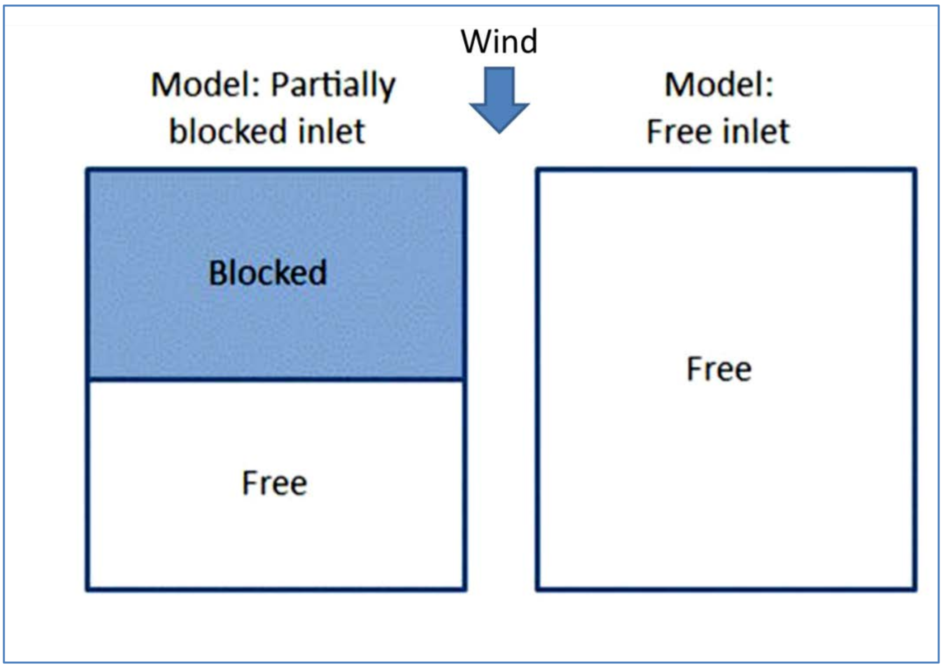

Figure 3: Modeling of a partially blocked entrance and an ideal free inlet condition.

3 RESULTS

After approx.1000 time steps, the flow inside the chimney was stabilized, with residuals below the chosen tolerance of 1e-8 for each time step. For no wind conditions the gases and smoke plume developed reasonably outside the chimney.

Reynolds number in the chimney is 7e5, based on diameter and mean velocity; Rayleigh and Grashof numbers based on the chimney wall length, mean flow temperature and velocity, reached values of 2.1e10 and 2.9e10 respectively. It must be pointed out, nevertheless, that the flow within the chimney is driven by forced convection (blowers outside the domain).

Numerical simulation of the partially blocked situation, with no wind, highlighted three main problems within the system chamber-chimney:

• Cyclonic flow in the chimney, generating helycoidal streamlines where the velocity vector inclination is beyond acceptable limits for measurements.

• Flow acceleration in the sector opposite to entrance, due to the effect of the lateral inlet and the deviation that gas suffers after impinging the wall

• Generation of a horizontal vortex in the pre-entrance chamber, due to obstructions in the inlet ducts. This vortex increases both problems mentioned before. These problems are illustrated in figure 4. The ideal situation of free gas inlet ducts eliminated the horizontal vortex in the chamber, as shown in figure 5, but the problems of cyclonic flow and local accelerations still remained unsolved.

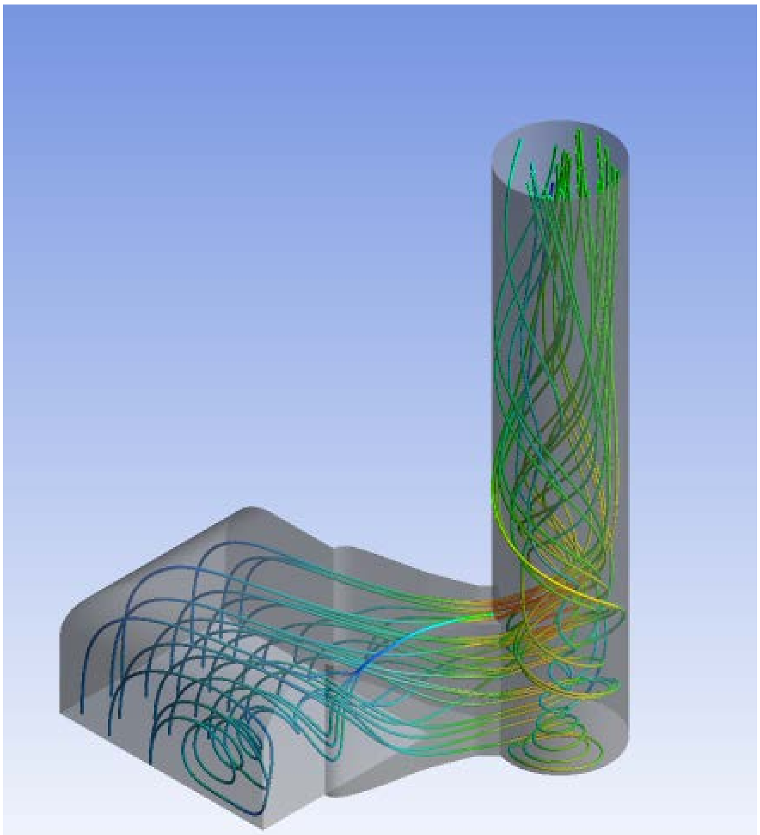

Figure 4: Path lines in the pre-entrance chamber and the chimney.

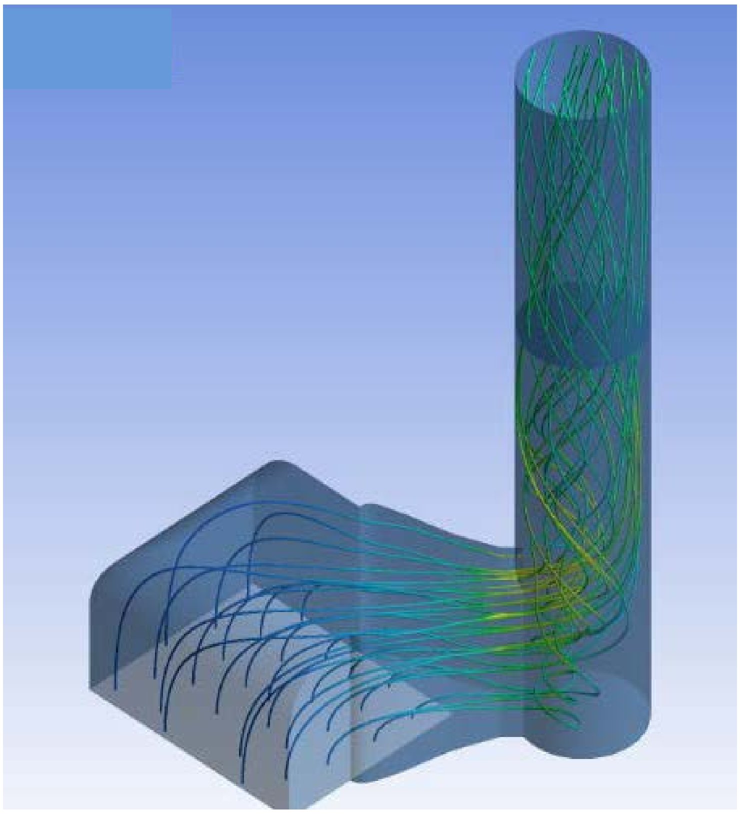

Figure 5: Path lines in condition of free inlet.

In order to reduce the cyclonic components in the chimney, a straightener device was designed, with the geometry shown in figure 6. A short flat plate on the lower side helps to attenuate the vortices before the flow passes through the device.

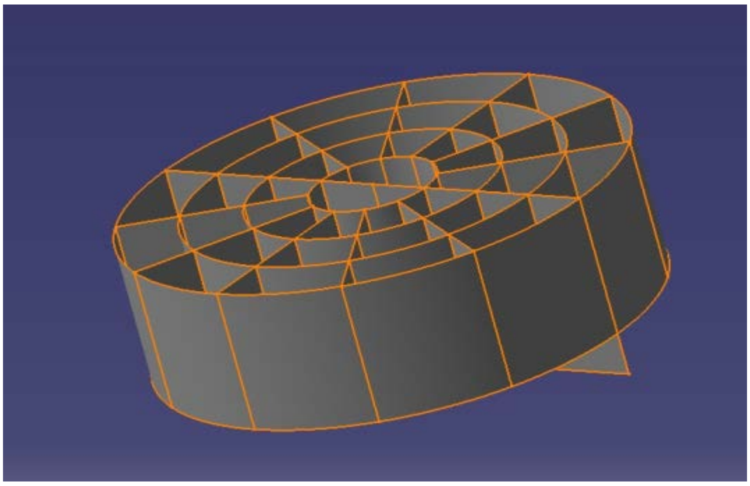

Figure 6: Proposed flow straightener.

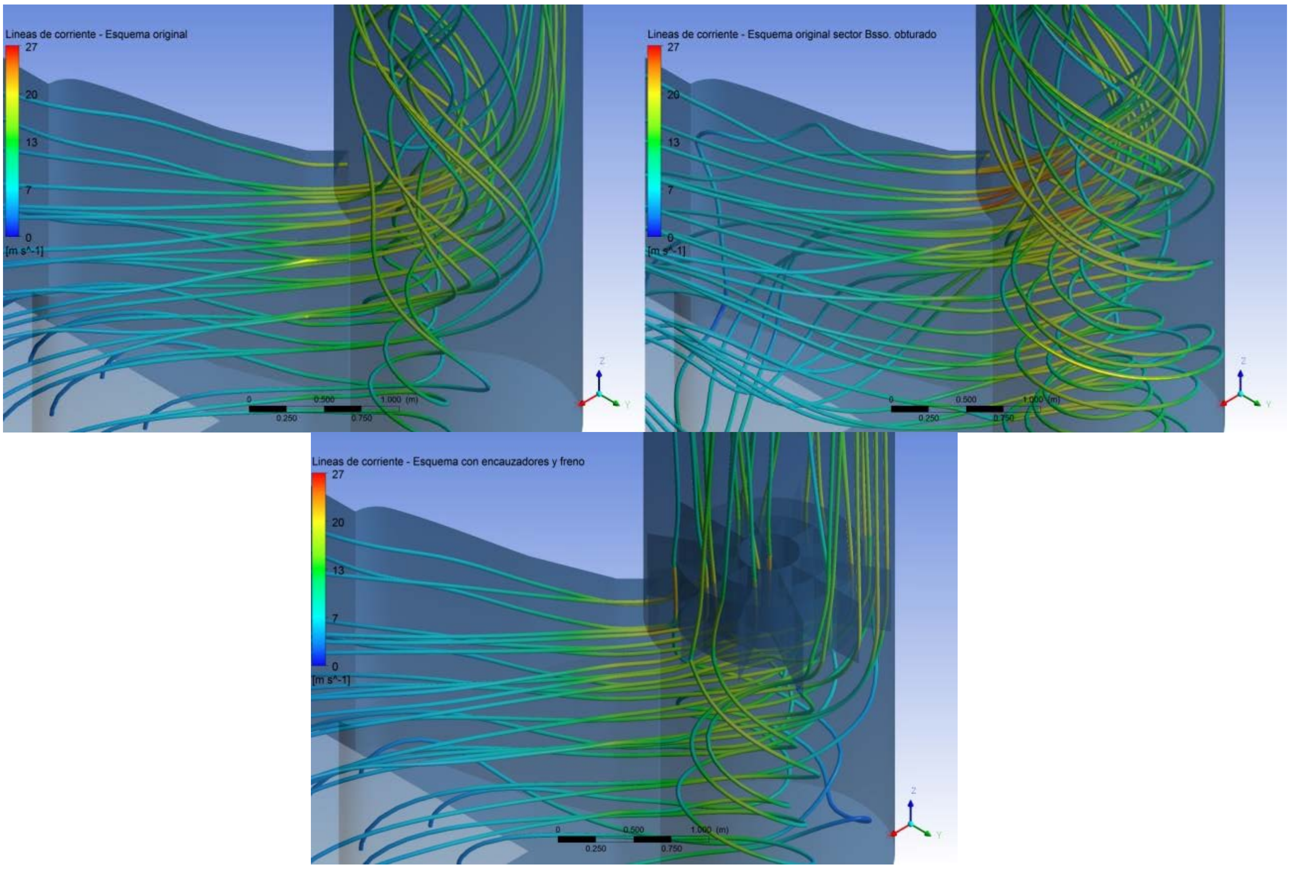

Figure 7: Flow configuration (pathlines) a) free inlet; b) partially blocked inlet; c) free inlet and flow straightener.

Figure 7 shows the effects on the flow at the chimney entrance of the free and partially blocked entrance ducts and of the flow straightener, for outside wind speed zero. Eliminating the longitudinal vortices increases the vertical velocity near the wall opposite to the chamber. This problem has not been solved yet. Nevertheless, a non-uniform velocity distribution can be consistently measured as far as the flow direction is uniform, which is the main goal achieved with the proposed straightener.

The inclination of the velocity vector in the chimney is reduced with this straightener from near 40 degrees to less than 5 degrees.

A measure of the cyclonic component in the flow is the parameter called “helicity”, *H* (Moffat, 1969, Saffman 1992). Its definition is:

$$H = \frac{(\vec{\nabla}\times\vec{V})\cdot\vec{V}}{\left|\vec{V}\right|} \tag{1}$$

The helicity can be interpreted as the projection of the vorticity $(\vec{\nabla}\times\vec{V})$ in the direction of the flow velocity $(\vec{V})$, being a quantitative indication of cyclonic flow. Cyclonic flows in ducts present then a high level of helicity.

The computation of vertical helicity in the chimney (figure 8), performed by ANSYS package, show its drastical reduction after the straightener. The effect of its geometry is to concentrate helicity in small cores, that dissipate shortly after entering the chimney.

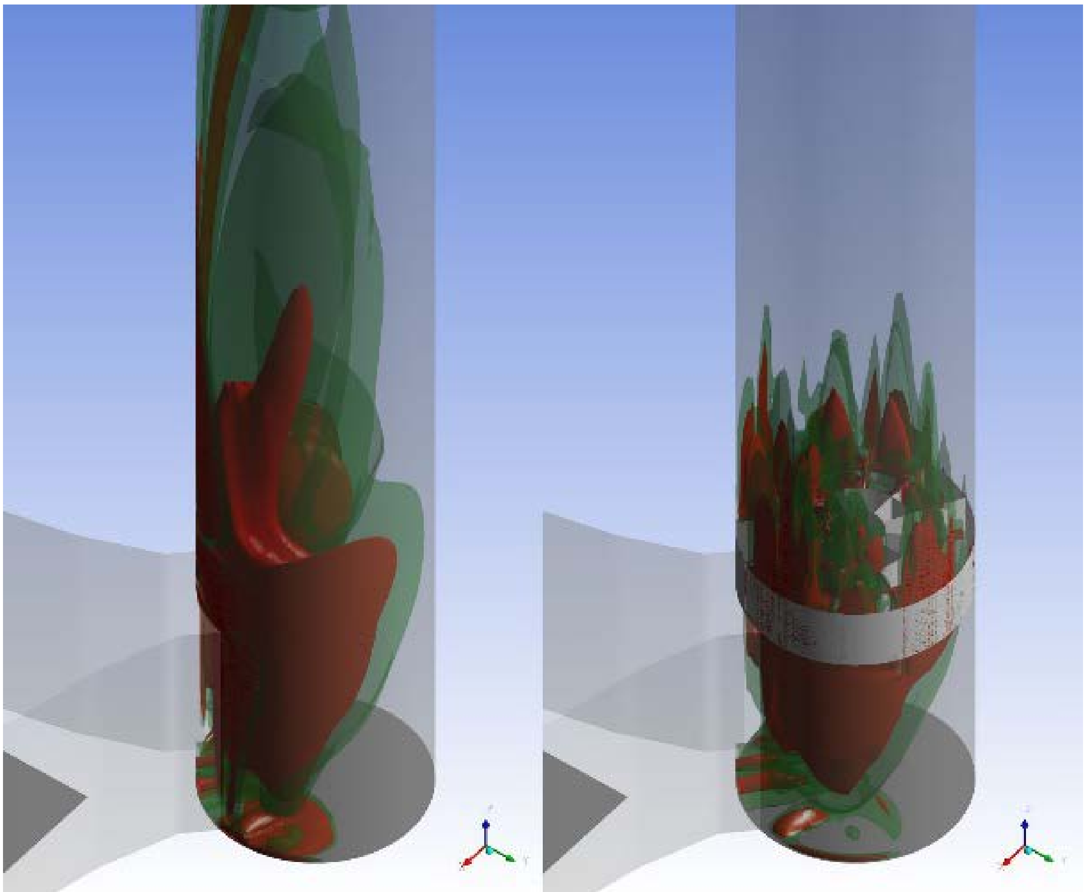

Figure 8: Isosurfaces of vertical helicity. Green is positive (vorticity pointing upwards), red is negative.

The question remains about how much pressure loss this flows straightener introduces. Since the pressure just before the straightener is highly non-uniform, the way of estimating the total pressure loss was to compare the total (static plus dynamic) pressure difference ΔP between the flow inlet to the chimney and the reference plane, at a distance of 2 diameters from the chimney top, for flow with and without the flow straightener. The difference in the results was attributed to the straightener. Non-dimensional loss coefficients K were then obtained by dividing these pressure differences by the dynamic pressure obtained with the average gases density ρ and velocity at the chimney $V_{ch,}$, as:

$$K = \frac{\Delta P_{fl.st.}}{0.5 \rho V_{ch}^2} \qquad (2)$$

The total pressure loss was computed for three cases: partially blocked inlet-no straightener, free inlet-no straightener and free inlet with straightener. The difference between the last two is the pressure coefficient attributed to the straightener. The first case is included for comparison purposes. Table 1 shows these results.

Case	K_{tot}	$K_{fl.\,str.}$
Partially blocked inlet – no straightener	1.54	
Free inlet	0.57	
Free inlet + straightener.	1.09	0.52

Table 1: Loss coefficients

Results show that the flow straightener introduces a loss factor of 0.52 in the system, which is comparable to that of a rounded elbow or other more sophisticated commercial flow straightener systems (Westfall) designed for minimum head losses. On the other hand, a partially blocked inlet introduces energy losses twice as high, but the system is still able to operate with acceptable efficiency, which implies that the introduction of the flow straightener, keeping the inlet clean, will not affect the chimney operation.

4 SYNTHESIS AND CONCLUSIONS

The evaluation of the flow in an assembly entrance chamber-chimney led to the following results:

- The geometry of the assembly originates vortex systems within the chimney, which prevent the flow to fulfill requirements of uniform velocity and direction. Cyclonic flow difficults accurate velocity measurements for the determination of mass flow.

- When the inlet ducts to the entrance chamber are partially obstructed, a horizontal vortex develops before the chimney inlet, that accelerates the flow in the upper region of the entrance and increases the longitudinal vortices strength in the chimney.

- Keeping the inlet ducts free by adequate maintenance improves the flow quality but does not solve the problem of inclined flow direction in the chimney, due to a couple of counterrotating vortices.

In order to solve this situation, a flow straightener design was proposed, with the following guidelines:

- To reduce or eliminate the cyclonic flow in the chimney;
- To achieve this goal keeping the velocity distribution as uniform as possible in the reference plane;
- To achieve this goal without introducing important energy losses in the flow;
- The solution should be simple, economic and of easy installation and maintenance.

The proposed flow straightener could reduce the maximal streamline inclination at the control plane, situated one diameter below the chimney outlet, from 35^o-40^o to less than 5^o. Its pressure loss coefficient, based on the gas mean velocity and density in the chimney was K = 0.52. At present, we are not satisfied with the velocity distribution obtained in the chimney reference plane, but simple modifications are being studied, which contribute to make the flow more uniform.

It must be pointed out that at present the system has been studied in conditions of no external wind and wind of 10 m/s and its operation was not affected by this variable.

REFERENCES

Andreozzi A., Buonomo B., Manca O., Thermal and fluid dynamic behaviors in symmetrical heated channel-chimney systems. *International Journal of Numerical Methods for Heat & Fluid Flow* 20 No. 7, 811-833, 2010.

Harris D. J., Helwig N.,Solar chimney and building ventilation, *Applied Energy* 84 135–146, 2007.

IRAM 29230 Método 1 A-EPA. Emisiones de fuentes estacionarias. Muestras y velocidad transversales para fuentes estacionarias con chimeneas o conductos pequeños.

Kazansky 1 Kazansky S., Dubovsky V., Ziskind G., Letan R., Chimney-enhanced natural convection from a vertical plate: experiments and numerical simulations. *International Journal of Heat and Mass Transfer* 46 497–512, 2003.

Moffatt, H.K.: The degree of knottedness of tangled vortex lines. *J. Fluid Mech*. 35 (1969), pp. 117–129.

Saffman, P. G.: Vortex Dynamics, Cambridge University Press 1992.

US EPA, CFR 40, http://www.epa.gov/foia/foiaregs.htm.

http://westfallmfg.com/3000_conditioner/