Available online www.jsaer.com

***Journal of Scientific and Engineering Research*, 2015, 2(2):79-93**

Research Article

ISSN: 2394-2630
CODEN(USA): JSERBR

Computational Fluid Dynamic Analysis of a Heater Chimney with and without a Flow Straightener

Ana. Scarabino[†]*, Federico. Bacchi[†], Ricardo J. Filace[‡] and Marcela Raviculé[+]

[†]Grupo de Fluidodinámica Computacional GFC, Departamento Aeronáutica, Facultad de Ingeniería, Universi-dad Nacional de La Plata., La Plata, Argentina
[‡] Departamento de Ingeniería de Procesos, Gerencia de Servicios Técnicos, Complejo Industrial La Plata, YPF SA, La Plata, Argentina
[+]YPF Tecnología S.A.

Abstract International standards for control of pollutants and particulate flow into the atmosphere require periodically monitoring of gases in discharge chimneys. For acceptable control, the streamlines must be straight, inclined by no more than a few degrees from vertical. 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. This work presents a numerical analysis of the flow within and outside a chimney of low length/diameter ratio with a lateral inlet for combustion gases, the gases dispersion in the atmosphere, the problematic flow configuration detected and a flow straightener proposed to reduce the vorticity within the chimney, which reduces the streamlines deviation from 35-40^o to less than 5^o.

Keywords chimney, cyclonic flow, flow straightener.

1. Introduction

Fuel gases from burning of fossil fuels have been on the focus of environmental discussions for much of the past decades [1]. This has led to a constant evolution of legal and regulatory aspects by the States; measurement systems, control and management policies by the industries; as well as public culture and awareness about environmental issues derived from gaseous effluent discharge to the atmosphere.

The evolution of environmentally-related measuring and sampling techniques has been simultaneous with the increasing exigencies over contaminant concentration levels on gaseous effluents. As a result, many industrial facilities had to be modified in order to fulfill increasingly strict environmental objectives.

In petrochemical industries, steam generators are typically among the systems with greatest impact on the discharge of gases into the atmosphere. Whether continuous emissions monitoring or periodically sampling techniques are being used, the need of reliable and accurate measurements is of great importance. Thus, not only the selection of appropriate instrumentation is required but the installation and properly location of sampling probes are of great concern.

For example, velocity measurements in stacks are usually performed by means of pitot tubes. Although the resulting experimental data may be reliable and accurate, the method is very sensitive to sensor misalignment with the flow. To minimize that, the sampling plane should be located at some specified distances downwards and upwards of the immediate flow disturbances (elbows, entrances, discharges, etc.). The flow in this plane must be as uniform as possible and the streamlines 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 [2-3].

Although many international and local standards describe the procedures and methodologies for selecting an appropriate sampling location on a duct, sometimes it is useful to determine if the current sampling location in existing equipments is suitable for a particular instrument and method.

The availability of computational resources has led to an increasing number of numerical studies performed to optimize the designs of all types of chimneys and ducts for fluid transport [4-6].

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 under certain operating conditions. 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 over 35 degrees to less than 5 degrees, meeting the criteria stated in environmental regulations for particle emission control [7-8] 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.

The numerical study of the actual situation also provided a means to validate the model, comparing predicted with measured values of gas velocity at the control plane, showing an acceptable agreement.

2. Methods

A. Geometry and measurements

Figures 1 and 2 show the chimney geometry and the location of the reference plane, where velocity and pollutants measurements take place. Gas velocities were only measured along the perpendicular lines A2 and A3, traversing the whole chimney diameter on each line. All dimensions in figure 2 are referred to the chimney inner diameter D.

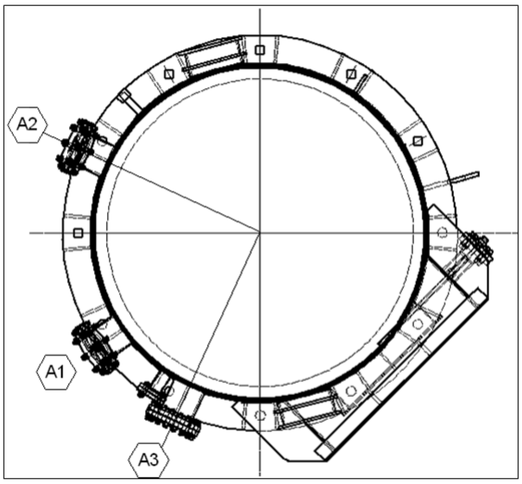

Figure 1: *Measurement points and lines A1 (not used), A2 and A3*

In order to evaluate the flow quality at the reference measurement plane, the flow velocity was measured following the procedures stated in US EPA, CFR 40, Method 2: Determination of Stack Gas Velocity and Volumetric Flow Rate (Type S Pitot Tube) [7]. This method is not accurate for direct measurement in cyclonic or swirling gas streams. When unacceptable conditions exist, alternative procedures, subject to the approval of the Administrator, must be employed to produce accurate flow rate determinations. An example of such alternative procedures is to install straightening vanes.

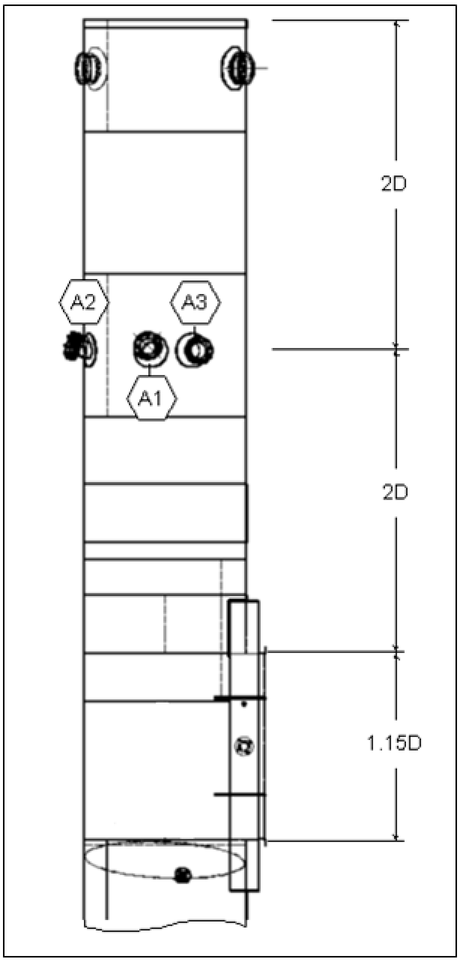

Figure 2: *Sketch of the chimney dimensions*

Due to confidential procedures, only one set of data (acknowledged as representative) was available for validation of the CFD (computational fluid dynamics) results. It was nevertheless recognized that measurements in the chimney central core showed a large variability, although this was not quantified. Results obtained with this methodology, although not conclusive, led to suspect the existence of cyclonic flow.

It was therefore decided to perform numerical (CFD) studies of the actual operating condition in order to get a detailed description of the flow motion, and also to propose and study possible modifications that could improve the velocity distribution in the chimney

B. Numerical simulation

The computational domain defined for this study 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 (Fig. 3). 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 analyzing the influence of wind in the chimney operation and the plume dispersion (Fig. 4).

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.

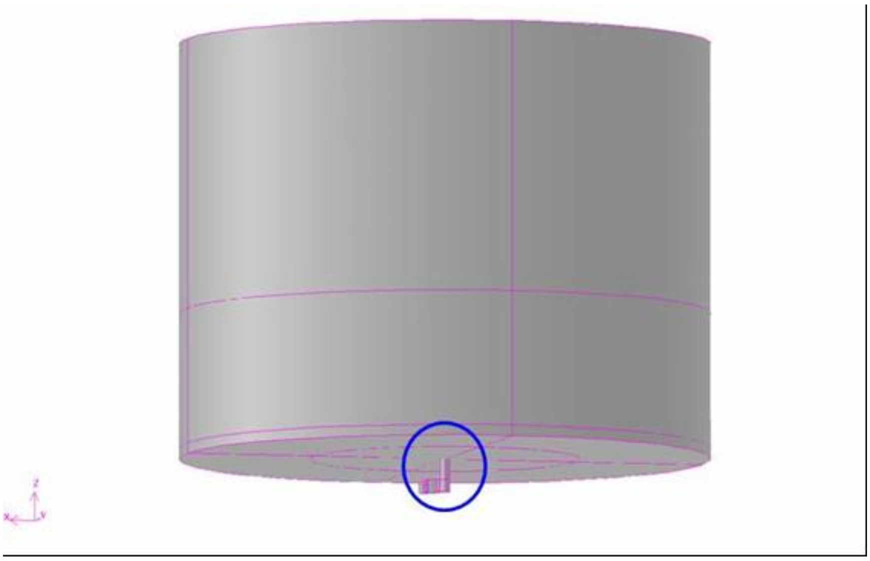

Figure 3: *Computational domain. The chimney is in the blue circle*

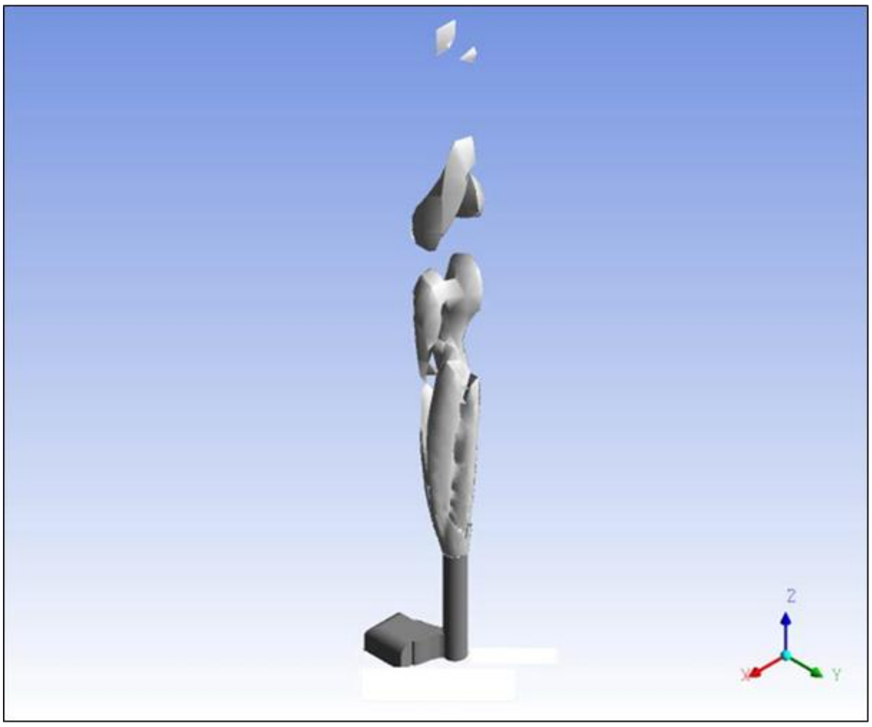

Figure 4: *Isosurface of gases concentration 90%.*

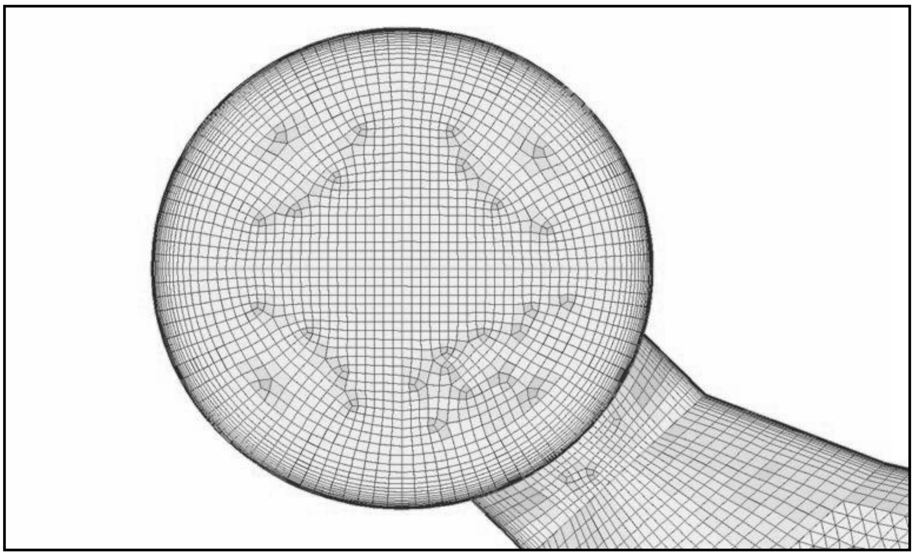

Figure 5: *Detail of mesh refinement in the chimney.*

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-ε turbulence model was used, with "enhanced wall treatment" [9]. The mesh was locally refined until it included 4e6 elements, in order to achieve grid-independent results. Figure 5 shows local mesh refinements. The time step for convergence was 1e-5 s.

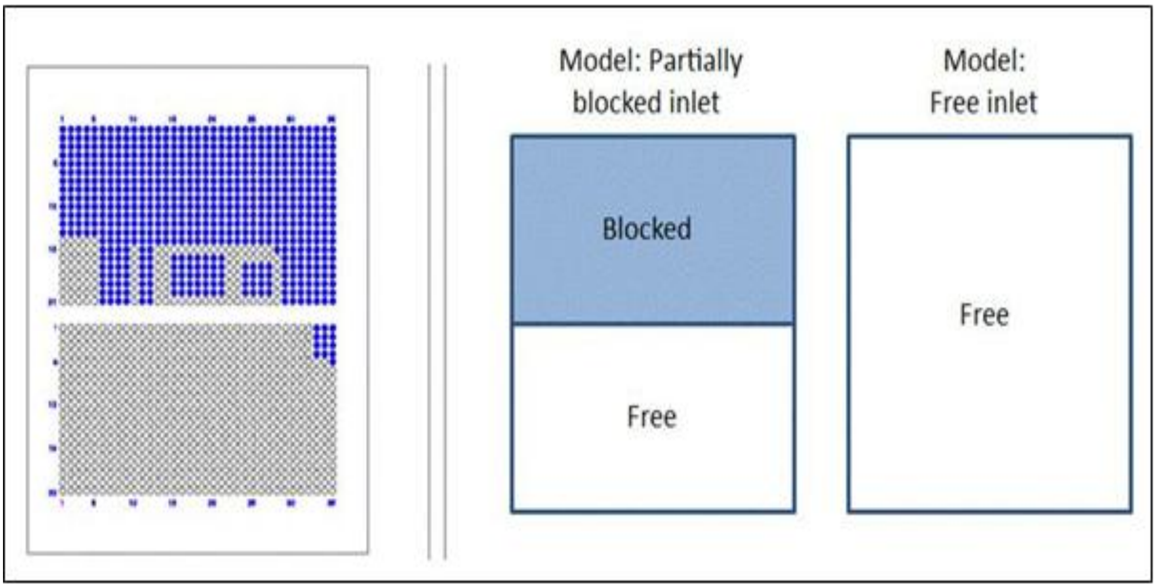

Figure 6: *Example of a partially blocked entrance, and modeling of this and the ideal free inlet condition*

Specified boundary conditions were “no slip walls”, mass flow, pressure and temperature at the chamber inlet and “pressure far field” (constant atmospheric pressure with specified temperature) for the atmosphere boundary. Wind velocity was set to zero. A reference plane was defined at a distance of 2D to the chimney exhaust for the analysis of local velocity and other variables distribution, and comparisons with experimental measurements.

The flow inlet to the chimney lateral pre-entrance chamber consists of a large number of ducts. After a period of operation, deposits of residuals obstruct the ducts near the chimney. The system still operates with acceptable efficiency, but the flow within the chimney is affected by this condition at the entrance. The available measured data correspond to this operating condition, therefore, it was necessary to simulate this case in order to validate the numerical model.

In order to model the partial obstruction of the entrance ducts, the inlet was divided in two sections as shown in Fig. 6. The operating conditions for the chimney are then “free inlet” and “partially blocked inlet”. In both cases the total mass flow is the same.

It must be pointed out that convection within the chimney is primarily forced by blowers outside the computational domain. Reynolds number at the chimney is 8e5.

3. Results

A. Validation and diagnose

After approx. 1000 time steps, the flow inside the chimney was stabilized, with residuals under the established threshold of 1e-8 for each time step. For no wind conditions, the gases and smoke plume developed reasonably outside the chimney, as shown previously in Fig.4.

Numerical results of vertical velocity distribution at the two measuring lines for the actual operating conditions matched reasonably the measured values, as shown in Fig. 7. The largest deviations between experimental and numerical results appeared in the central region, where measurements had a great variability and thus larger uncertainties. Due to confidentiality reasons, all velocities have been normalized dividing by the maximum local value, $Vref$, as

$$v^* = v/Vref$$

and a nondimensional traversing distance on each measuring line has been defined as

$$r^* = (x - R)/R$$

where R is the chimney radius, $D/2$, and x is the distance from the pitot entrance at the chimney wall for each measuring line, A2 and A3 respectively.

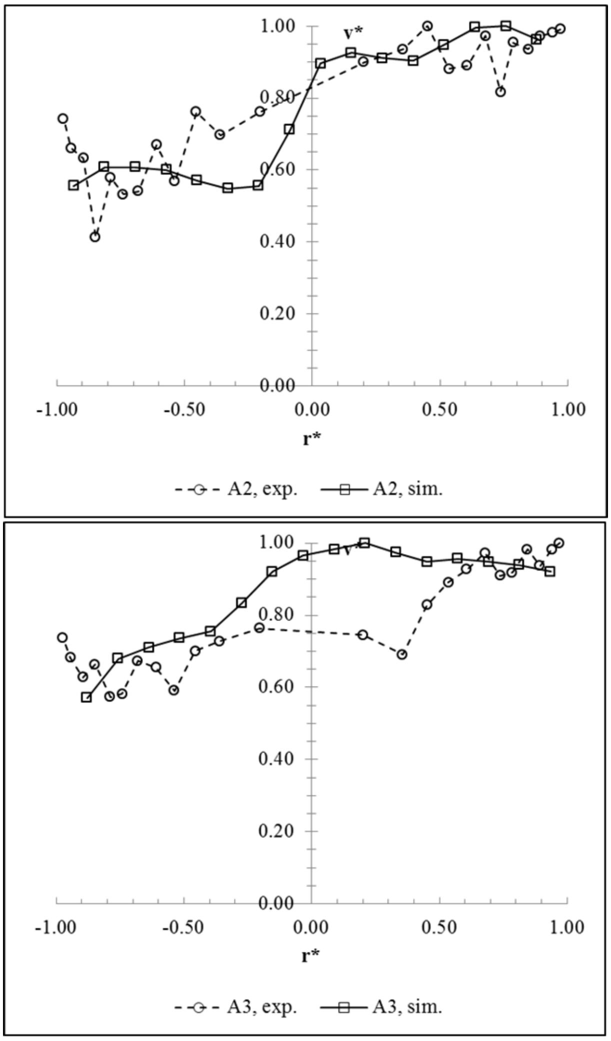

Figure 7: *Computed and measured vertical velocities at the measuring lines A2 and A3. Partially blocked entrance.*

Furthermore, numerical results gave a detailed description of the flow configuration, showing the existence of cyclonic flow originated at the chimney entrance, a condition that was suspected from previous measurements. 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 particle 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 Figs. 8 and 9.

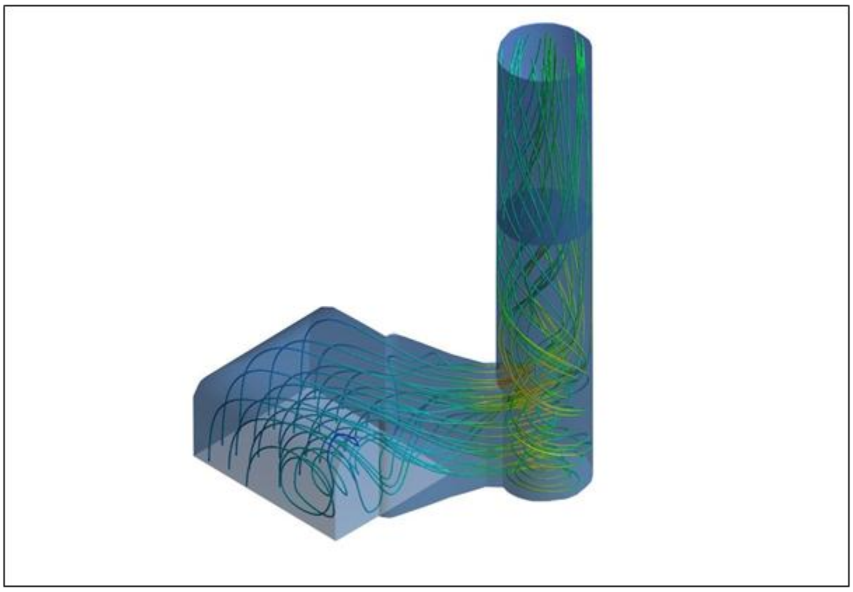

Figure 8: *Streamlines in the pre-entrance chamber and the chimney.*

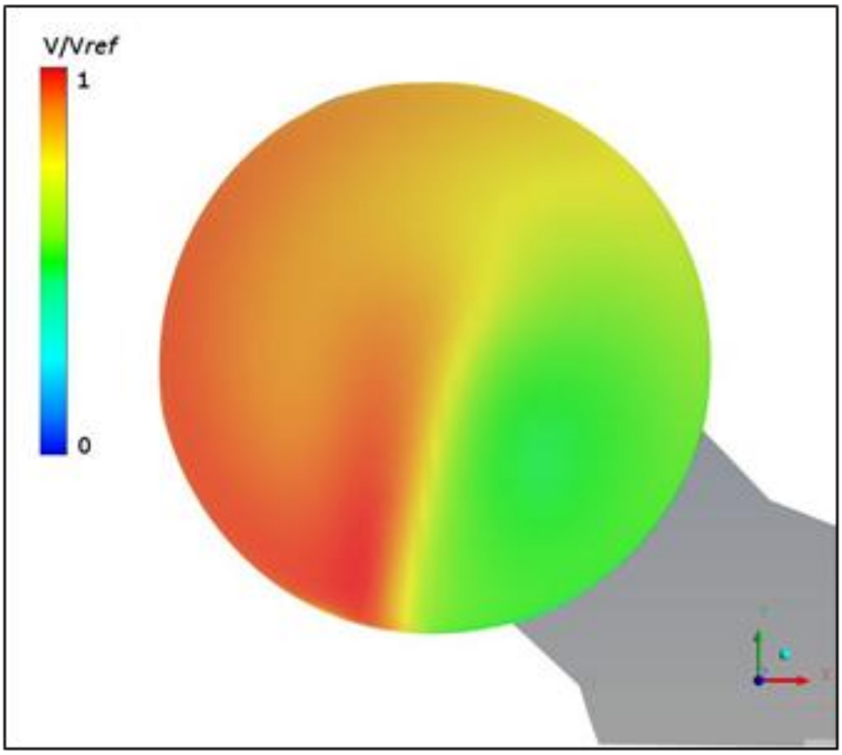

Figure 9: *Normalized vertical velocity distribution in the reference plane. Vrefis the maximum local value.*

B. Free operating condition

The ideal situation considering free gas inlet ducts, elim-inated the horizontal vortex in the chamber, as shown in Fig. 10, but the problems of cyclonic flow and local accelerations remained unsolved, as Fig. 11 clearly shows.

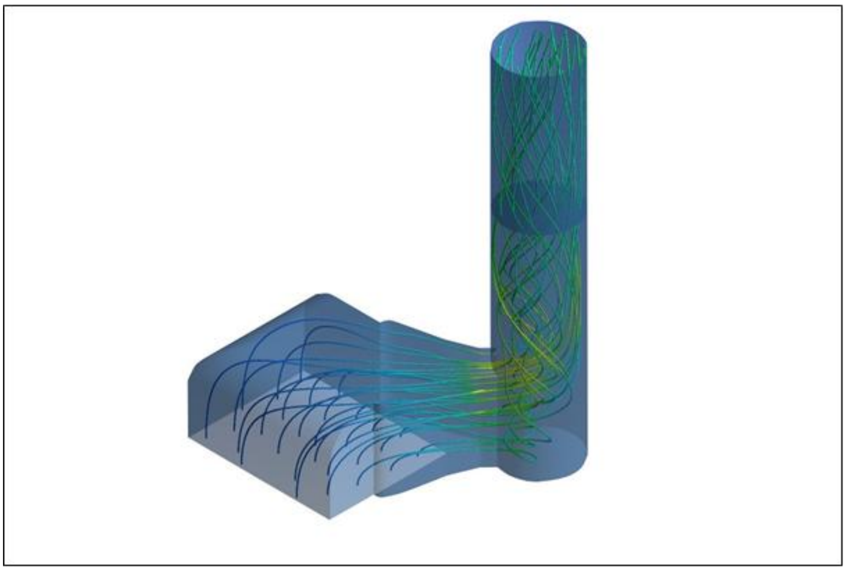

***Figure 10**: Pathlines in condition of free inlet.*

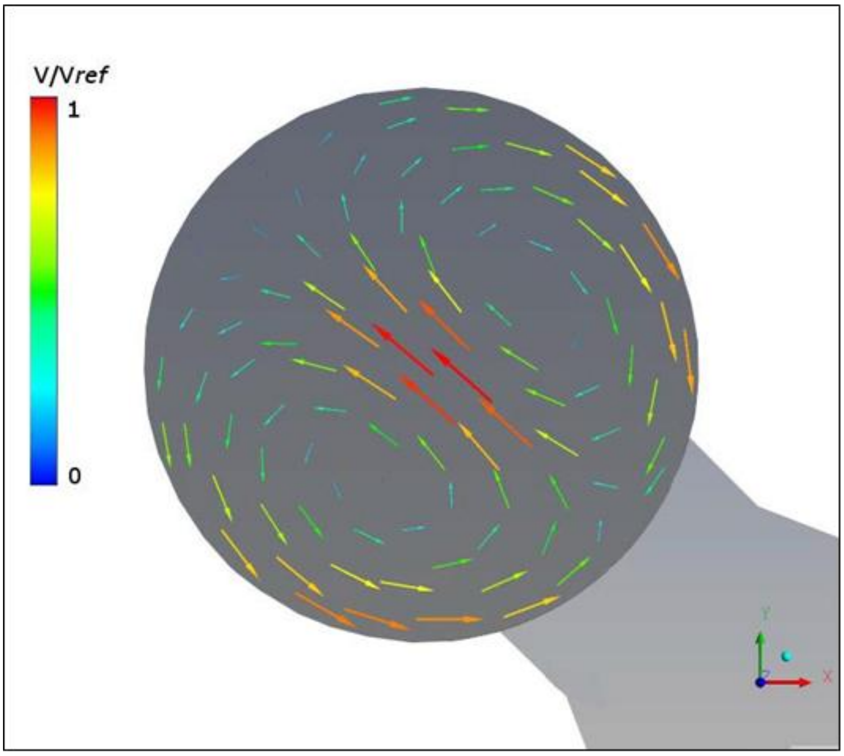

***Figure 11:** Horizontal components of the velocity at the reference plane. Vref is their local maximum.*

C. Flow straightener

In order to reduce the cyclonic components in the chimney, a straightener device was designed, with the geometry shown in Fig. 12. A short flat plate on the lower side helps to attenuate the vortices before the flow passes through the device.

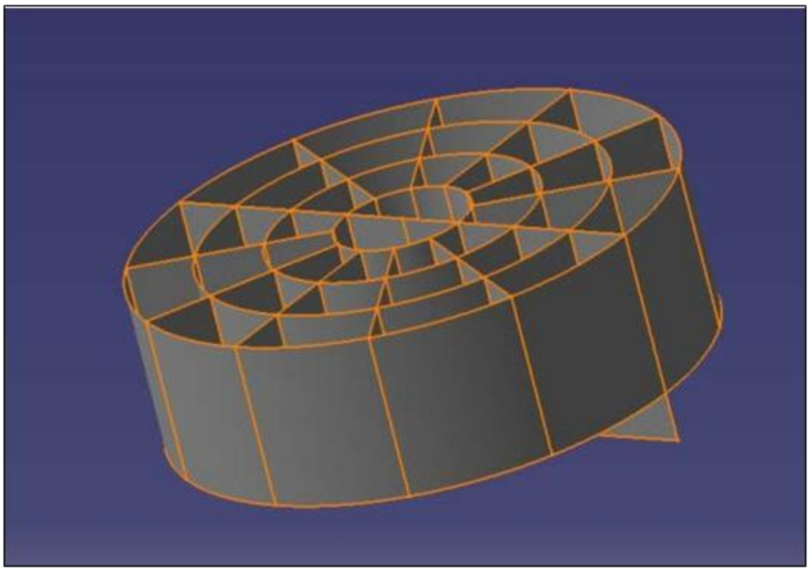

Figure 12: *Proposed flow straightener.*

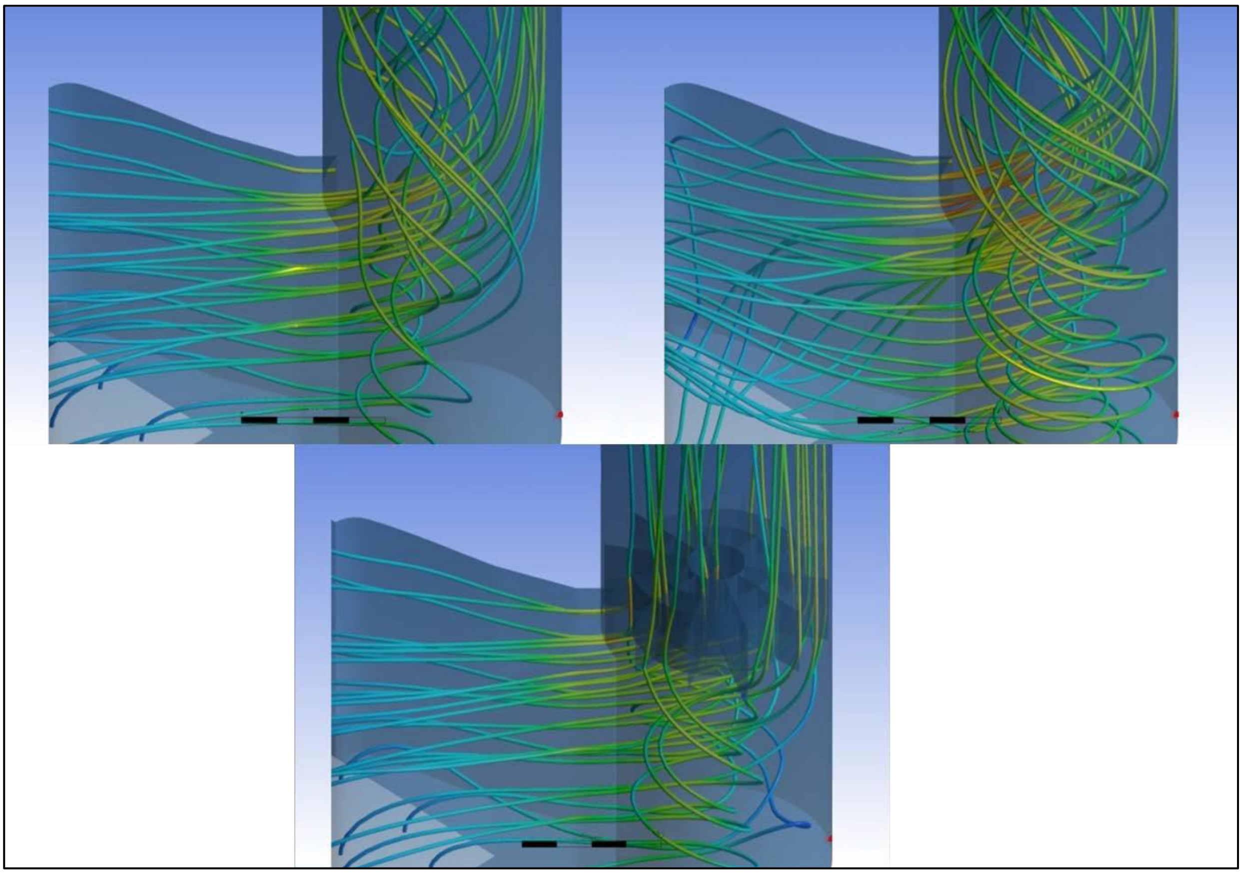

Figure 13: Flow configuration (pathlines). a) Free inlet; b) Partially blocked inlet; c) Free inlet and flow straightener

Figure 13 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. Figure 14 shows the flow in the system chamber-chimney, with and without the flow straightener and the velocity vectors at the reference plane.

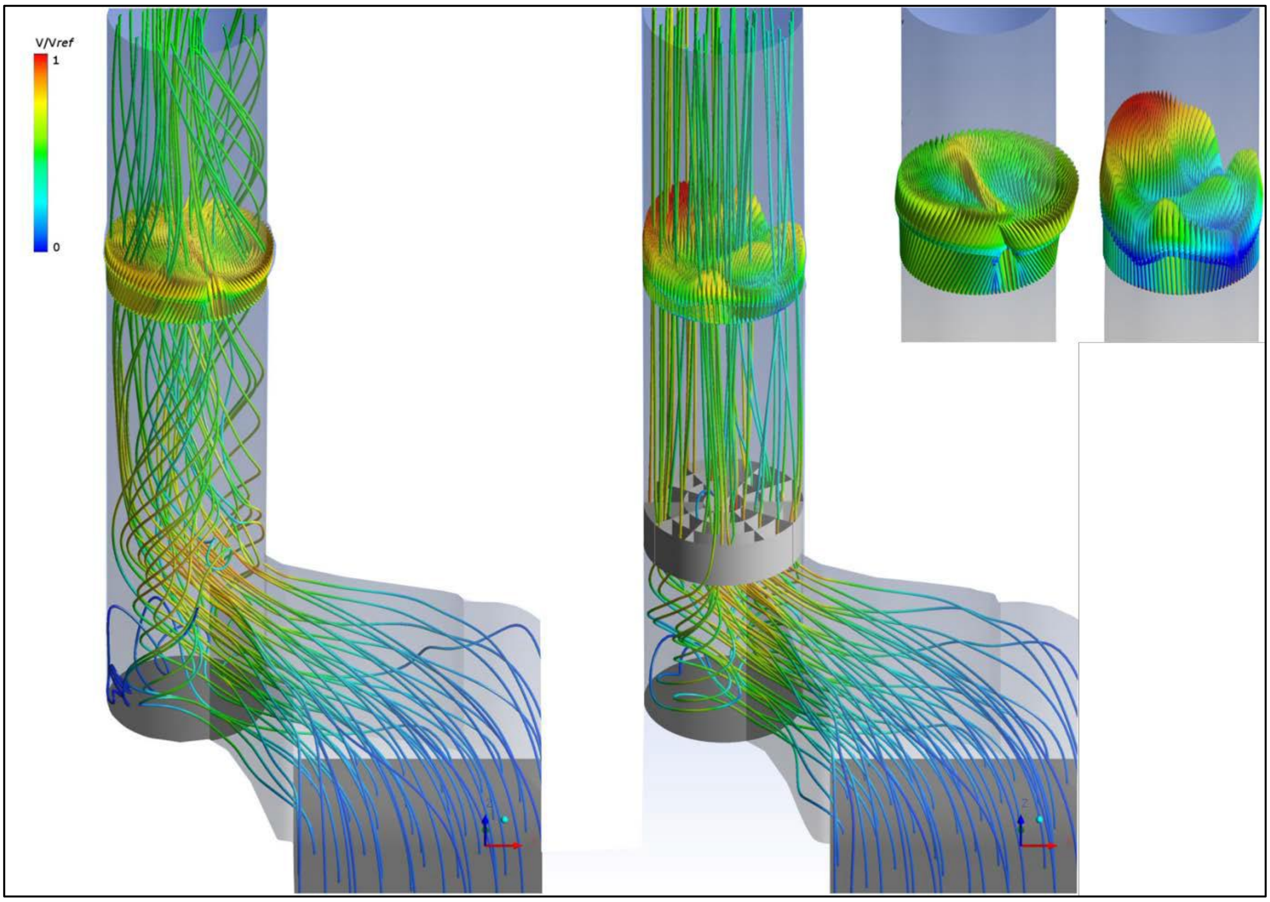

Figure 14: Pathlines with and without the flow straightener and velocity vectors at the reference plane. *Vref* is the maximal velocity at the reference plane with the flow straightener.

Concerning the velocity modulus, a consequence of reducing the cyclonic flow is the acceleration of the flow near the wall opposite to the lateral entrance. This condition, although not optimal, still allows accurate flow measurements if the adequate number of points is chosen. Further improvements in the straightener design are being considered at this time in order to obtain a more uniform velocity distribution at the reference plane.

The model was run in calm atmospheric conditions (no wind) and for SW wind speeds of 5 m/s and constant atmospheric temperature of 10 ^{o}C. No atmospheric boundary layer was simulated at this stage. The boundary conditions for this flow were set with 10% turbulence intensity and a turbulence reference length of 2 m. The wind velocity and orientation were adopted as representative for a typical breezy day in the chimney location. It was found that while the exterior flow is obviously influenced by the wind, the characteristics of the flow inside the chimney are not sensibly affected by it, at least for the studied velocity of 5 m/s.

D. Flow inclination

Figure 15 shows the velocity vector inclination with respect to the vertical, which should ideally be zero. It can be seen that this angle in the actual condition reaches 30 to 35 degrees in the chimney center, and it is over 40 degrees for line A2 near the wall. This problem is present for both the actual partially blocked operating condition and an ideal free-entrance operation. The proposed flow straightener reduces this inclination below 5 degrees, thus allowing Pitot tube measurements with misalignment errors below accepted criteria. Results are shown for condition of no external wind.

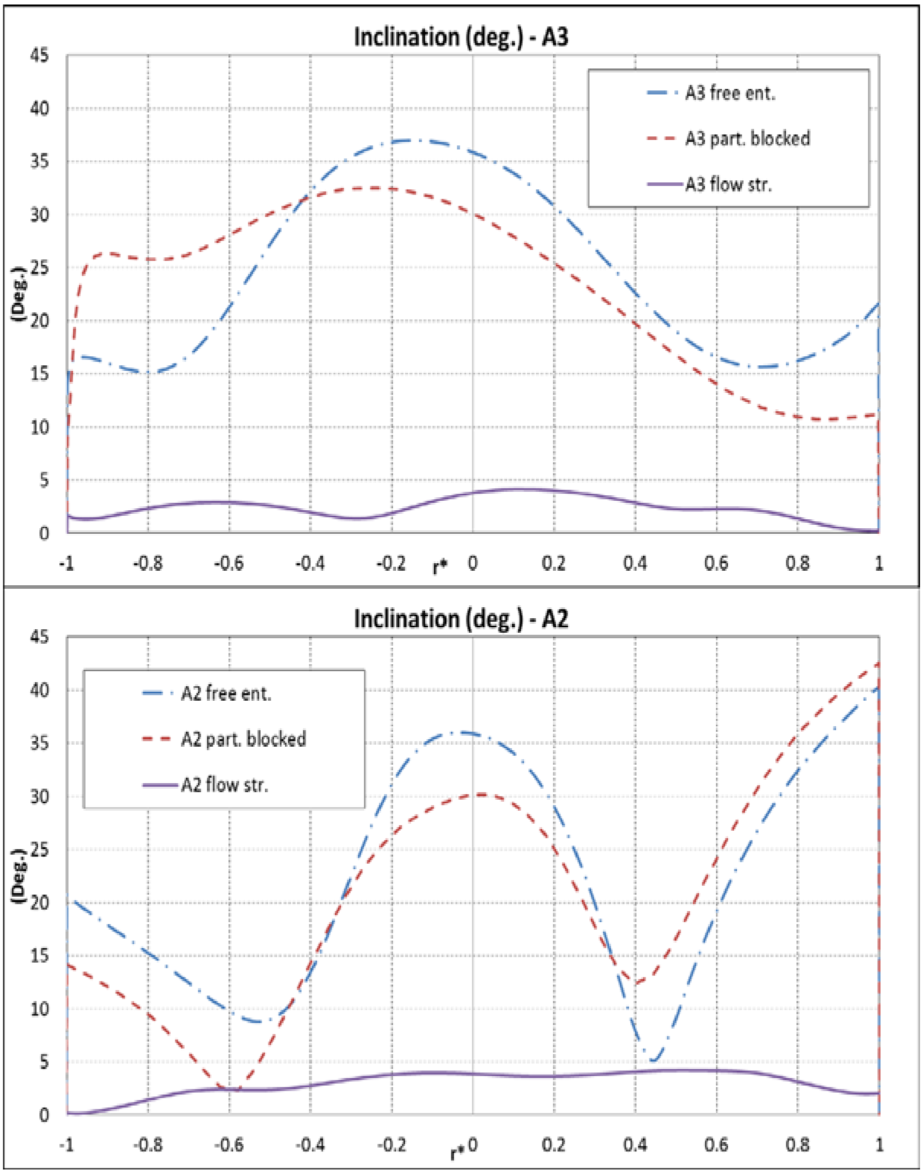

Figure 15: *Flow inclination along measuring lines A2 and A3 (no wind). r* = (x-R)/2R*

E. Helicity and turbulent dissipation

A measure of the cyclonic component in the flow is the parameter called “helicity”, *H*[10,11]. Its definition is:

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

The helicity can be interpreted as the projection of the vorticity in the direction of the flow, being a quantitative indication of cyclonic flow. Cyclonic flows in ducts present then a high level of helicity.

The computation of vertical helicity (projection of vorticity on the vertical component o velocity) in the chimney (Fig. 16), performed with Ansys CFD postprocessing package, show its drastical reduction after the straightener. The effect of its geometry is to concentrate helicity in small cores, which dissipate shortly after entering the chimney.

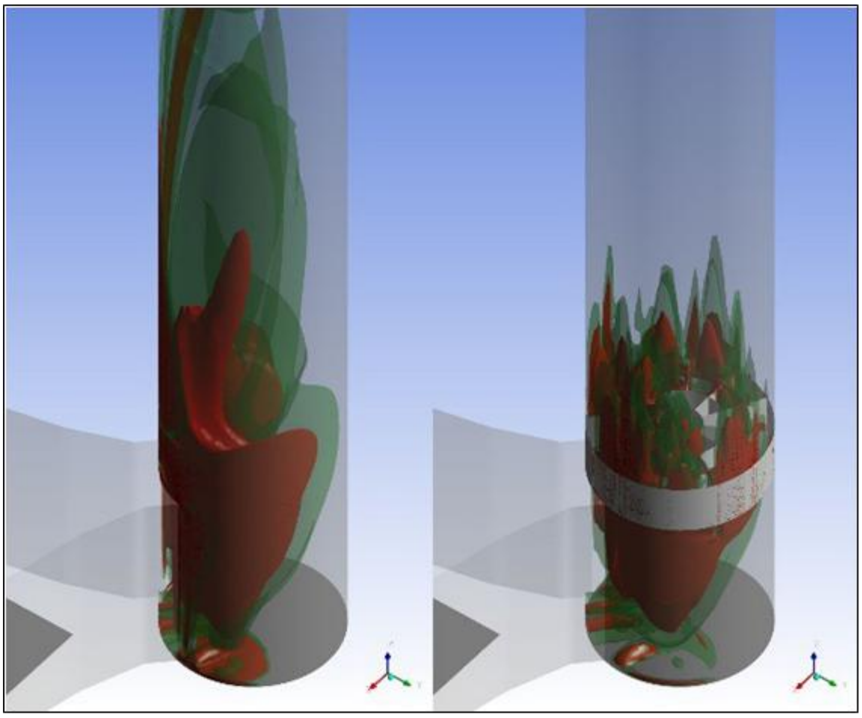

Figure 16: *Isosurfaces of vertical helicity. Green is positive, red negative*

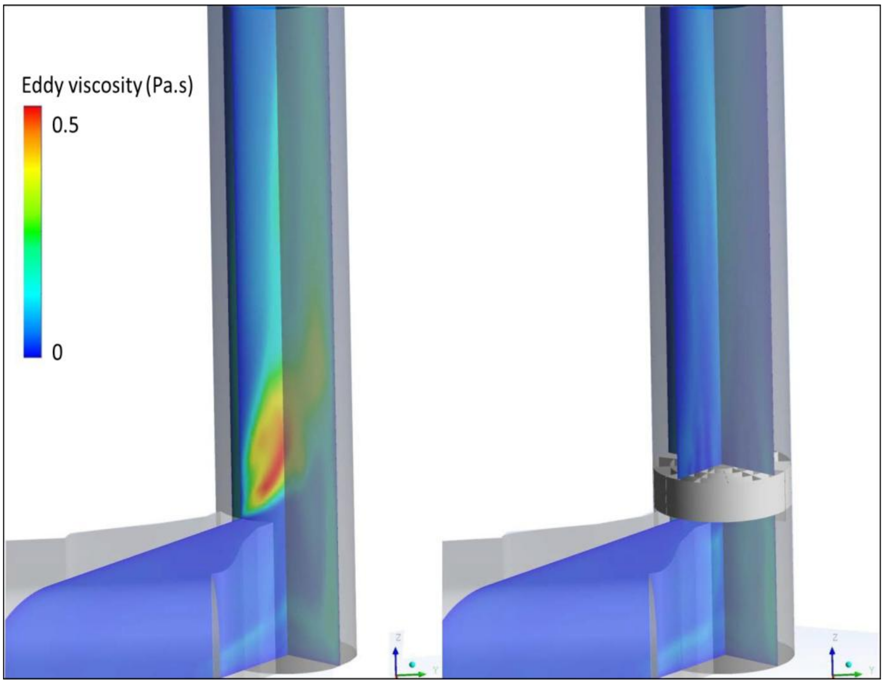

Figure 17: *Contours of eddy viscosity without and with flow straightener.*

The straightener not only reduced the helicity, but it also lowered the turbulence levels. Figure 17 shows contours of eddy viscosity, a measure of turbulent dissipation, in two perpendicular planes. The reduction introduced by the flow straightener is of one order of magnitude.

F. Head losses

The question remains about how much total 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 between the flow inlet to the chimney and the reference plane, at a distance of 2 diameters before the chimney top, for conditions with and without the flow straightener. The difference in the results was attributed to the straightener losses. Non-dimensional loss coefficients K were then obtained by dividing these total pressure differences by the dynamic pressure obtained with the average gases density and velocity at the chimney, as:

$$K = \frac{\Delta P_T}{0.5\,\rho_{ch}\,V_{ch}^2}$$

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 gives the pressure losses attributed to the straightener. The first case is included for comparison purposes.

Table 1: Loss coefficients

Case	K_{tot}	$K_{fl.\ str.}$
Partiallyblocked inlet– nostraightener	1.54	--
Freeinlet	0.57	--
Freeinlet+ straightener.	1.09	0.52

Results show that the flow straightener introduces a loss factor of 0.52 in the system, which is comparable, for example, to that of a rounded elbow or other more sophisticated commercial flow straightener systems (Westfall, [12]) 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, should not affect the chimney operation

4. Synthesis and Conclusion

The use of flow straighteners is common in industry. Nevertheless, their design and application for chimneys has not been exhaustively studied. This particular work, even covering only one specific heater chimney, highlights aspects of the flow, which are of common occurrence in this type of systems, and hints for a simple solution to the problem of cyclonic flow.

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 makes difficult to obtain 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, which 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 35-40 degrees to less than 5 degrees. 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 will contribute to make the flow more uniform.

The system was studied in conditions of no external wind and of wind of speed 10 m/s and its operation was not affected by this variable.

Cyclonic flow components were reduced at a level that should not affect velocity measurements at the reference plane 2D below the chimney discharge.

The simplicity of this solution makes it preferable over other options, as the construction of a new, longer chimney. Additionally, its installation has little to no impact in the operation, since its construction is external and the time required for placing the straightener inside the chimney is minimum, being this an operation that can be performed in practically any maintenance shutdown of the system.

Further work includes the experimental validation of the straightener efficiency, once it is constructed and installed, and the study of modifications leading to reach a more uniform velocity distribution at the measurement plane. In addition, it is planned to carry out a study of the plume dispersion for different wind velocities and atmospheric stability conditions.

References

1. Goldemberg, J., & Lucon, O. (2010). *Energy, environment and development*. Earthscan.
2. Smith, C. R., & Hopper, P. B. (1997). *The use of cyclonic flow removal tabs to minimize CEMS flow measurement errors* (No. CONF-970145--). Air & Waste Management Association, Pittsburgh, PA (United States). Pp. 1112-1116.
3. Smith, C. R., Yordy, E. L., & Hopper, P. B. (1995, October). Elimination of cyclonic flow in chimneys to meet CEMS criteria. American Power Conference, Chicago, IL (United States).
4. Kazansky, S., Dubovsky, V., Ziskind, G., & Letan, R. (2003). Chimney-enhanced natural convection from a vertical plate: experiments and numerical simulations. *International Journal of Heat and Mass Transfer*, *46*(3), 497-512.
5. Harris, D. J., & Helwig, N. (2007). Solar chimney and building ventilation. *Applied Energy*, *84*(2), 135-146.
6. Andreozzi, A., Buonomo, B., & Manca, O. (2010). Thermal and fluid dynamic behaviors in symmetrical heated channel-chimney systems. *International Journal of Numerical Methods for Heat & Fluid Flow*, *20*(7), 811-833.
7. US EPA, CFR 40 Part 60, available at http://www.epa.gov/.
8. IRAM 29230 Método 1 A-EPA. Emisiones de fuentes estacionarias. Muestras y velocidad transversales para fuentes estacionarias con chimeneas o conductos pequeños. (In Spanish).
9. Ansys, A. F. (2011). 14.0 Theory Guide. *ANSYS inc*, 218-221.
10. Moffatt, H. K. (1969). The degree of knottedness of tangled vortex lines. *Journal of Fluid Mechanics*, *35*(01), 117-129.
11. Saffman, P. G. (1992). *Vortex dynamics*. Cambridge university press.
12. http://westfallmfg.com/3000_conditioner/.