Study and set-up of a wind tunnel for assessing ammonia emissions from the field

Poster · June 2015

7 authors, including:

Ester Scotto di Perta
University of Naples Federico II

Boccia Lorenzo
University of Naples Federico II

Salvatore Faugno
University of Naples Federico II

Collins Okello
Gulu University

AIIA 2015 International Mid-Term Conference
Italian Society of Agricultural Engineering – Naples, June 22-23, 2015

Study and set-up of a wind tunnel for assessing ammonia emissions from the field

E. Scotto di Perta[a,b], L. Boccia[a], S. Faugno[a], C. Okello[c], G. Sorrentino[d], M. A. Agizza[d], S. Pindozzi[a,]*

[a] Department of Agriculture, University of Naples Federico II, Via Università 100, Portici (NA)
[b]Department of Agriculture, Forestry, Nature and Energy (DAFNE), University of Tuscia, Via S. Camillo de Lellis s.n.c., 01100 Viterbo
[c]Department of Biosystems Engineering, Gulu University, P. O. Box 166, Gulu, Uganda
[d]Department of Chemical, Materials and Production Engineering, University of Naples Federico II, P.le Tecchio 80, Napoli

Introduction

The ammonia volatilization process depends on many factors (soil conditions, agricultural practices, slurry characteristics and climate conditions, especially the wind speed and rainfall). Consequently it is difficult to assess the atmospheric ammonia fluxes and there is no standard measurement for this purpose. The wind tunnel is often preferred for ammonia emission assessment, because it is more flexible and simple to use.

Few studies have been carried out to determine, which aerodynamic conditions are established inside the tunnel and how these one could influence the emission rate.

The aim of this study is to set up a wind tunnel, which has a better aerodynamic performance, and to investigate the effects of the wind tunnel geometry and flow behavior using CFD simulations and wind speed measurements.

Materials and Methods

- ### Geometry design

It is characterized by a main chamber, situated between a divergent and a convergent duct (respectively expansion and contraction section). The bottom of the main chamber is open and facing the emission surface, in particular the experimental area was 40 cm wide and 80 cm long, so it consists of a 0,32 m^2. Different wind tunnels sizes were used and compared to understand their effects on ammonia emission measurements. The dimensions of the tunnels are listed in details in the Table 1.

Table 1. Wind tunnels and their design dimensions

Parameter	A25-25	B50-15	C50-25	D50-50	E50-25-160	F50-25-25
WT width (m)	0,40	0,40	0,40	0,40	0,40	0,40
WT height (m)	0,25	0,15	0,25	0,50	0,50	0,25
Main chamber length (m)	0,80	0,80	0,80	0,80	1,60	0,80
Divergent length (m)	0,25	0,50	0,50	0,50	0,50	0,50
Convergent length (m)	0,25	0,50	0,50	0,50	0,25	0,25

- ### Computational fluid dynamics (CFD)

CFD technique have been used to simulate ventilation performance, by means of the Commercial software Fluent 15.0. The k-ω SST turbulence model was applied in this study and validated with the experimental results.

Velocity test and smoke test were performed, to define the aerodynamic condition inside the tunnel. The measurements were performed using a portable anemometer (HTA 4200, TEKKAL), with a measuring range between 0,3 to 40 m/s and a resolution of 0,01 m/s.

Three cross-sections were chosen for the measurements, at the position of 1/8 (x=10 cm), 1/2 (x=40 cm), 7/8 (x=70 cm) of length of the main chamber, respectively. Each one was characterized by 24 measuring points.

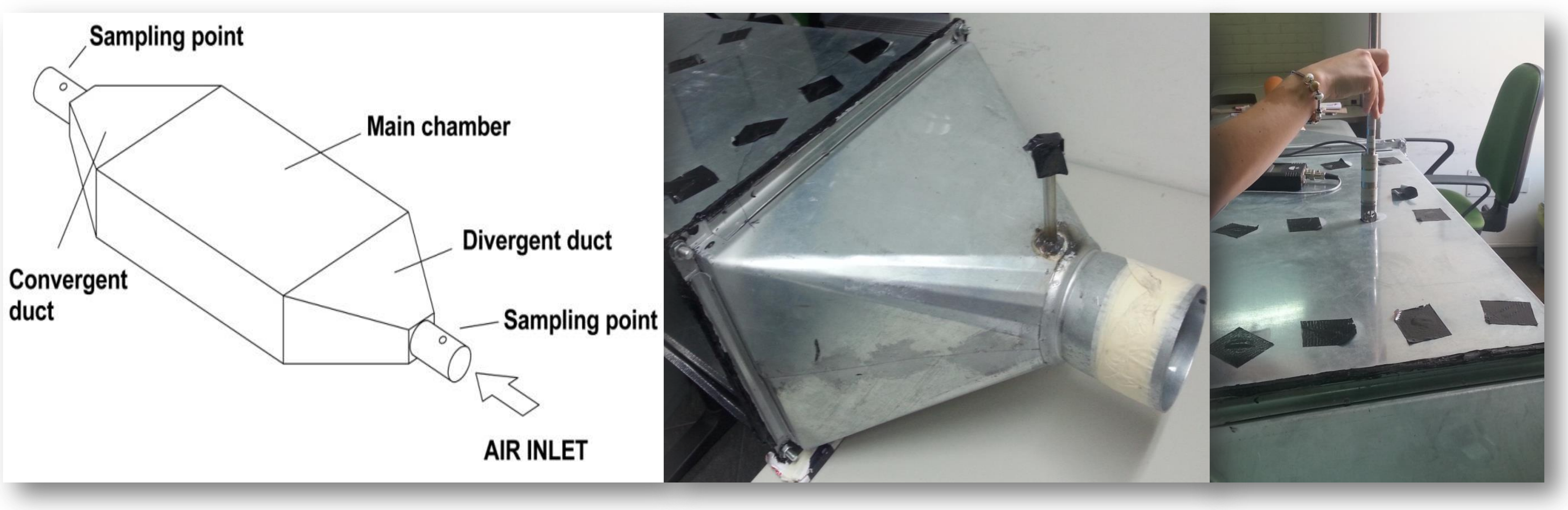

Figure 1. Schematic drawing of the wind tunnel, a detail of the divergent diffuser and velocity test measurements

Results and discussion

- ### CFD Validation

Figure 2 shows the comparison of velocity profiles between CFD simulations and measurements of two tunnels: A25-25 and F50-25-25. Despite possible measurement errors, due to the accuracy of the anemometer, and numerical calculations errors associated with simulations, results confirm the validity of the chosen turbulence model. Both results show the strange asymmetric behaviour of the flow pattern, located in the divergent diffuser and a big recirculation zone developed inside the chamber.

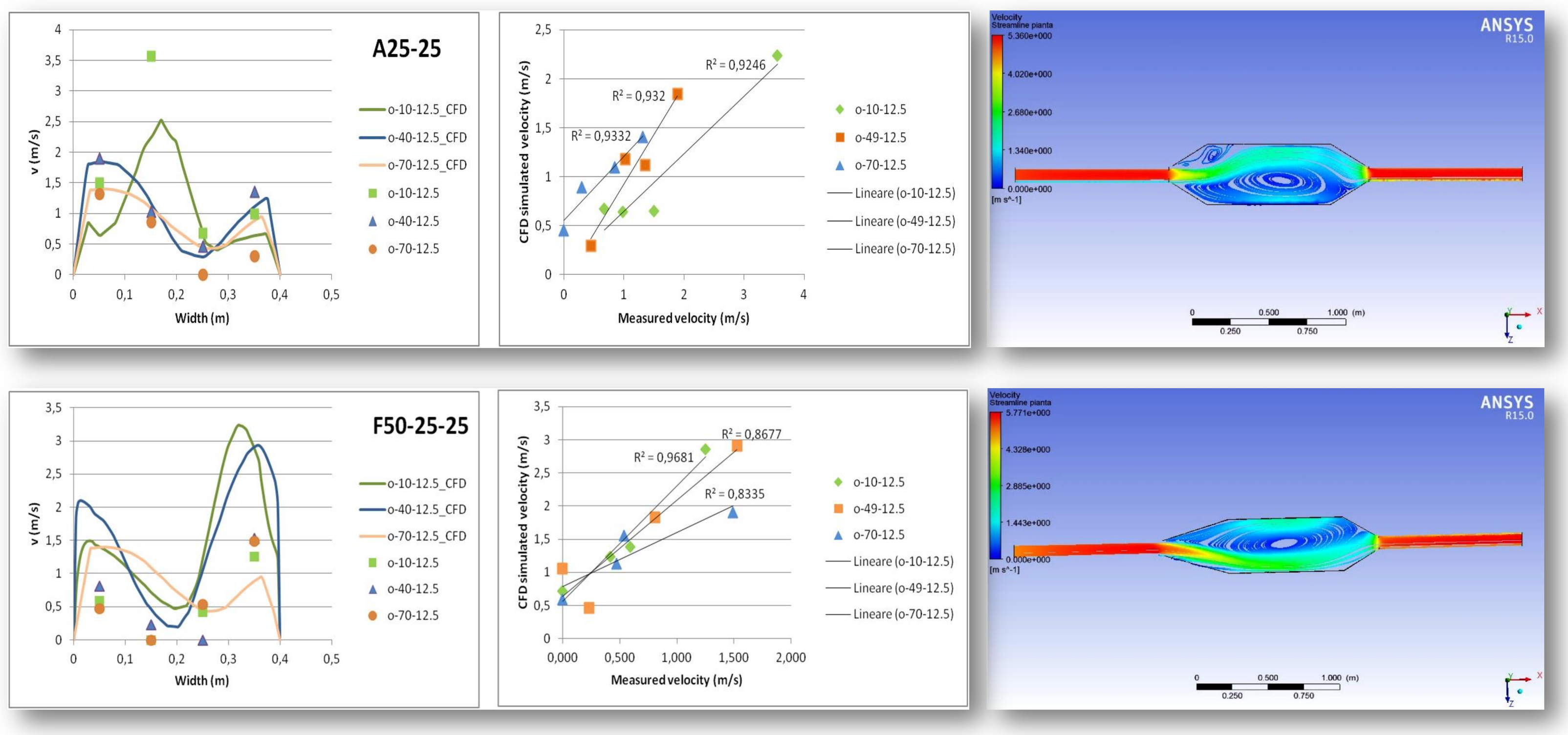

Figure 2. From the left: comparison of velocity profiles between CFD simulations and measurements of two tunnels: A25-25 and F50-25-25, velocity streamlines predicted by CFD simulations.

- ### Flow behaviour

In laminar condition, the flow in a symmetric chamber configuration with a plane expansion is almost always symmetric. It becomes asymmetric at a critical Reynolds number and would persist up to turbulent Reynolds numbers. In these cases, an asymmetric mean flow pattern is observed depending on the shape of the channel, the dimension of expansion section and the flow Reynolds number. The asymmetry of the steady flow is due to asymmetry-breaking bifurcation and the larger separation zone may occur on either side of the channel without any preference. This flow asymmetry is sometimes attributed to the so called "Coanda effect" (Durst et al., 1974; Jiang et al., 1995; De Zilwa et al., 2000; Nabavi, 2010).

- ### Improvement of geometry

The phenomenon of a flow that passes through the hood, generating a rotation region inside the chamber itself, is called "jet effect". It becomes more significant if the width of the tunnel is bigger then the height. In order to reduce this effect, a tunnel with an increased length have been investigated (E50-25-160), as suggest by Jiang et al., 1995.

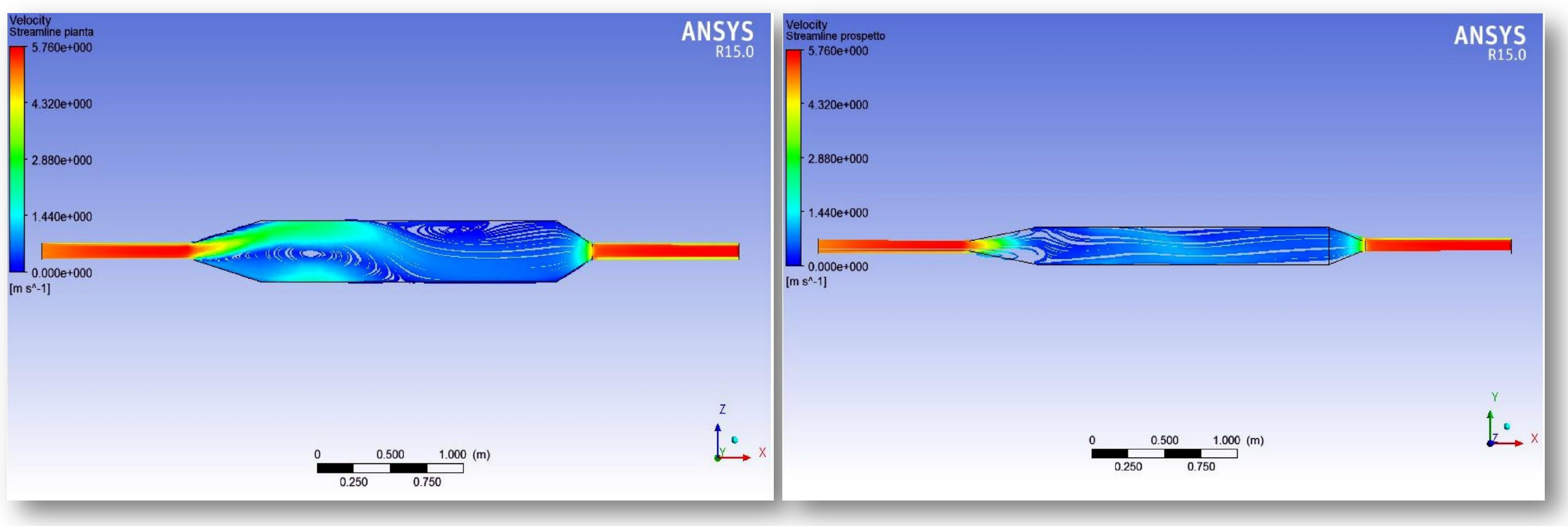

Figure 3. Velocity streamlines predicted by CFD simulations F50-25-25 configuration with a chamber with a double length

As shown, two unequal recirculation zones appear: the larger separation region occurs in the first part of the tunnel and then on the other side, in correspondence of the inversion of the flow. All the rotations occur around a vertical axis. As can be seen, a higher distance from the inlet reduces the inlet expansion effect, so that is possible to obtain a stable flow .

Conclusion

- All of experimental measurements have been made in air suction conditions, because preliminary results of velocity test demonstrate that the wind tunnel works better.
- The observed flow asymmetry could affect the turbulent mixing and the ammonia transfer rate, therefore uniform flow distribution inside the tunnel should be obtained. On the other hand, the space limitation of the chamber to be easy transportable, requires short working length. Thus, uniform flow is something that needs to be achieved with the installation of flow distribution devices.
- Finally, it has to be noted that although the CFD simulations give a good idea of the air flow behaviour inside the tunnel, numerical calculations always have an error associated with them. For this reason, experimental and literature studies should be made to have a sufficient knowledge about the reproduced phenomena.

Stefania Pindozzi: stefania.pindozzi@unina.it
Department of Agricultural Science University of Naples Federico II
Tel +39 081 2539128

Acknowledgments: *This research was realized under the Project " Optimizing the use of manure as a resource" funded by the Agriculture Department of the Campania Region.*