DESIGN AND CONSTRUCTION OF A NEW BOUNDARY LAYER WIND TUNNEL

Antonio NAVARRO MANSO
PhD. Civil Engineer
Hydraulic Engineer Area, University of Oviedo
Assistant Professor
navarroantonio@uniovi.es

Ricardo UCHE GARCÍA
Civil Engineer
Hydraulic Engineer Area, University of Oviedo
Master of Civil Engineering Student
uo232541@uniovi.es

Nicolás VALLINA PITA
Civil Engineer
Hydraulic Engineer Area, University of Oviedo
Master of Civil Engineering Student
uo231036@uniovi.es

Eduardo ÁLVAREZ ÁLVAREZ
PhD. Mechanical Engineer
Hydraulic Engineer Area, University of Oviedo
Assistant Professor
edualvarez@uniovi.es

Rodolfo ESPINA VALDÉS
Civil Engineer
Hydraulic Engineer Area, University of Oviedo
Associate Professor
espinarodolfo@uniovi.es

Eva MARTÍNEZ GARCÍA
Mining Resources & Energy Engineer
Exploitation & Mining Area, University of Oviedo
Master of Civil Engineering Student
uo267991@uniovi.es

ABSTRACT

Hydraulic Engineering Area at University of Oviedo has built a new Wind Tunnel (WT). The aim of this paper is to describe the design and construction of this outstanding facility, much bigger and better than the other four WT made years ago. A relevant and innovator aspect is that our students participate actively in the process, which is an excellent complement to their background and to the theoretical knowledge they achieve during the course titled Wind Engineering. Facing real problem of wood structure assembly, they also develop basic skills about aerodynamic concepts associated with aerodynamics issues.

Wind tunnels are devices that enable engineers to study the flow over objects of interest, forces acting on them and their interaction with the flow. Typically, values of pressure and velocity at any point in the flow, and forces on the structures can be solved.

Although the usefulness of Computational Fluid Dynamics (CFD) methods has improved over time, thousands of hours of Wind Tunnel Tests (WTT) are still essential for the development of a new aircrafts, wind turbines, suspension bridges, tall buildings or any other design that involves complex interactions with the flow.

It is well known that analytical solutions of the equations that predict the behavior of flows (i.e. Navier-Stokes equations) can’t be solved completely but simplifying models and applied boundary conditions. Consequently, growing interest of many branches of industry and science in low speed aerodynamics, and due to the persistent incapability of achieving accurate solutions with numerical codes, Low speed Wind Tunnels (LSWTs) are essential and irreplaceable during research and designing stages.

The referred wind tunnel, designed for teaching purposes wind tunnel is placed at Polytechnic School of Mieres (EPM), although some research projects and student's competitions could be done as well. When using for research the choice of an appropriate equipment of measurement (force balance, Pitot tube, hot wire anemometers and scanner pressure), is therefore crucial in obtaining reliable and accurate measurements.

The general layout of the proposed wind tunnel is described as follow: test chamber, where the scale model is placed, is located at the outlet of the nozzle, just in the flow discharge area. Upstream of the test chamber we find the other two main components of the wind tunnel: the contraction zone and the settling chamber. The other crucial component is of course the power plant or fan. Between the fan and settling chamber there is the remainder of the components, the diffuser.

The main specifications of the facility are the dimensions of the test section, 2.25 m^2, and the desired maximum operating speed, 40-50 m/s. The flow quality, in terms of turbulence level and flow uniformity, must be specified in accordance with the applications. The circuit is formed by a closed ring of 31 m long. A great-enhanced performance of the wind tunnel is that the test chamber is 10 m long and it will allow us to study Atmospheric Boundary Layer phenomena (ABL).

KEYWORDS: Wind Tunnel, Computational Fluid Dynamics, Wood structures, Construction

1. Introduction and object

The purpose of this article is to present the design and construction project of a boundary layer wind tunnel at EPM, University of Oviedo, developed by the Hydraulic Engineering Area and the Fluid Mechanics Area, Department of Energy. The construction is being carried out by professors and students of the both Degree and Master in Civil Engineering.

The objective sought with WT construction is the elaboration of basic research and practices for students, focused on the study of the interaction between wind and civil structures: bridges, buildings, wind turbines, panels, roofs, etc. It is well known that in many occasions the technical standards belittle the effects of wind and in many cases these loads of such a dynamic nature are responsible for the design of exposed and flexible bridges, singular buildings, roofs, facades, etc.

The oversizing of certain aerodynamic effects is also frequent on the part of the regulations. That is why, in order to analyze these effects on structures, wind tunnels have been developed as a fundamental tool in the design process of these constructions, allowing the study of the prototype by means of scale models.

The inventors of the first flying machines were also the pioneers in the construction of primitive wind tunnels. At a certain moment they were aware that they needed to understand the physics of the problem they were facing and with that they built instruments to test their models. The first measurements of the aerodynamic forces, carried out in the seventeenth century, were based on a device called "whirling arm" (rotating arm), which rotated the test object around an axis, i.e., was the model the one that moved while the air remained static. This device generated too much turbulence due to the connection arm, and therefore, it was very difficult to determine the true speed between the model and the air. Nowadays, improving this relative framework, the paradigm is based on the model remaining stationary while accelerating a flow of air around it.

Wind tunnels can be of different types: aeronautical, low turbulence, smoke, automobile, climatic, boundary layer, etc.; in addition, they present different configurations, both open and closed. However, they all have one element in common: a powerful fan (or fan array) that is responsible for driving the air through the duct system. With all this, several techniques and measuring instruments are used to auscultate the model tested, seeking to know speeds, pressure distribution, aerodynamic forces and other flow variables.

The use of scale models in turn requires taking into account the laws of similarity, in order to be able to translate the experimental results to the real conditions. The requirements to extrapolate the experimental results to the prototype are based on the fulfillment of geometric, kinematic and dynamic similarities. Generally, it is impossible to comply with all the parameters involved in the phenomenon being studied, so in practice it is enough to ensure that certain dimensionless numbers are within a defined range, depending on the characteristics of the problem.

The experimental tests carried out in wind tunnels are also fundamental to validate numerical simulation techniques. This calibration is necessary to get appropriate results with finite volume models. It should be remembered that experimental tests are those that very reflect the reality presented in the interaction of the object with the wind.

2. Design

The general design of the proposed wind tunnel is shown in Figure 1. The air flow runs counter-clockwise. Upstream of the test chamber we find the two main components of the wind tunnel: the contraction and the settling chamber. The other crucial component is, of course, the power plant. The rest of the components only serve to close the circuit and minimize the loss of energy. However, the diffuser 2 and the contiguous curve design also have an important influence on the flow quality and are responsible for more than 50% of the total losses. In the section 2.1, main components of the wind tunnel and the design parameters chosen are described.

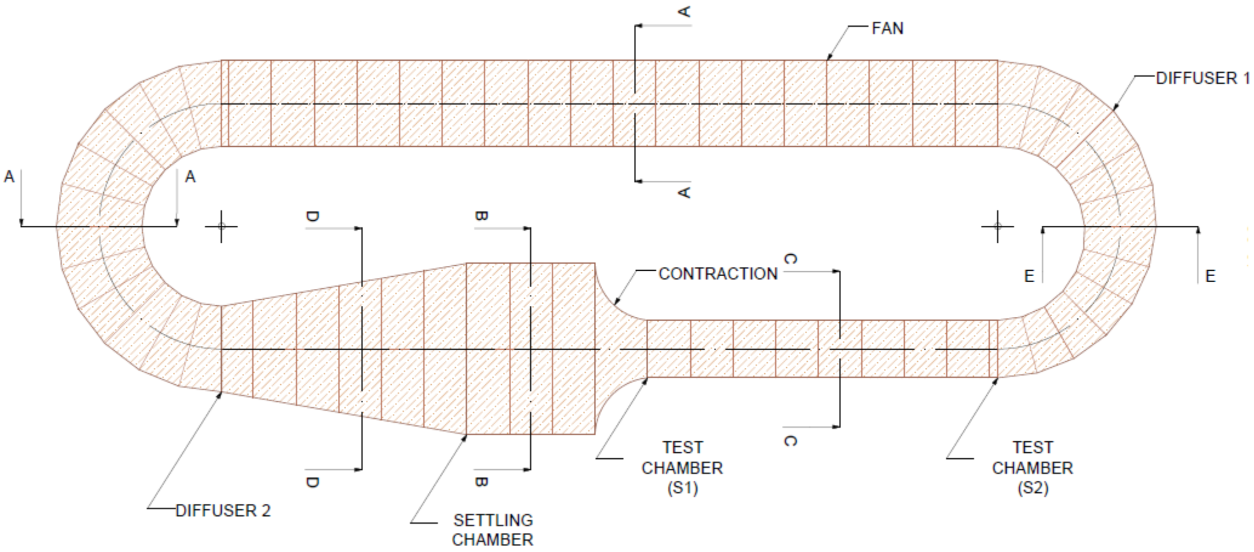


Figure 1. Plan of the EPM´s wind tunnel.

2.1. Test chamber

The size of the test chamber must be defined according to the wind tunnel main specifications, which also include the operating speed and desired flow quality. Test chamber size and operating speed determine the maximum size of the models and the maximum achievable Reynolds number, Re.

The cross-section shape depends on the applications. For civil or industrial applications, in most of the cases, a square or rectangular cross-section is recommended; the test specimens are usually bluff bodies and their equivalent frontal area should not be higher than 10% of the test chamber cross-sectional area in order to avoid the need of making non-linear blockage corrections. The EPM´s wind tunnel has a test chamber section of 1.5 m x 1.5 m and a length of 10 m, which allows carrying out tests of ABL (Atmospheric Boundary Layer).

A value of 50 m/s has been specified as the nominal speed at the entrance of the test chamber, which guarantees test power with Re greater than 10^5. Therefore, the flow rate that must circulate through the tunnel is (see expression (1)):

$$\mathrm{Q} = \mathrm{V} \cdot \mathrm{S} = 50\,\frac{\mathrm{m}}{\mathrm{s}} \cdot 1.5\ \mathrm{m} \cdot 1.5\ \mathrm{m} = 112.5\ \frac{\mathrm{m}^3}{\mathrm{s}} \qquad (1)$$

2.2. Contraction

The contraction or “nozzle” may be the most critical part in the design of a wind tunnel; it has the highest impact on the test chamber flow quality. Its aim is to accelerate the flow from the settling chamber to the test chamber, further reducing flow turbulence and non-uniformities in the test chamber. The flow acceleration and non-uniformity attenuations mainly depend on the so-called contraction ratio, N, between the entrance and exit section areas. Although, due to the flow quality improvement, N should be as large as possible, this parameter strongly influences the overall wind tunnel dimensions. Therefore, depending on the expected applications, a compromise for this parameter should be reached.

In our case, the main restriction for the settling chamber was the available space, so the chosen size was 4.74 m x 3.78 m section and 3.76 m long, which implies an N (2) of:

$$\mathrm{N} = \frac{A_S}{A_t} = \frac{4.74\ m \cdot 3.78\ m}{1.5\ m \cdot 1.5\ m} = 7.1 \qquad (2)$$

For the design of the contraction a logarithmic curve has been chosen (Figure 2). This curve has a good performance and solves the problems of lack of space and implies a greater constructive ease. To minimize the possible vortex that can appear in the angle of encounter between the settling chamber and the contraction, a chamfer at 45 ° has been depleted. The length of the nozzle is 1.5 m. The equations that describe the logarithmic curves used are presented below, both for the vertical plane (3) and horizontal plane (4):

$$\mathrm{y} = b_1 \cdot b_2 \cdot z[Ln(b_2 \cdot z) \cdot b_3] = 1.690 \cdot 0.627 \cdot z[Ln(0.627 \cdot z) \cdot 0.938] \qquad (3)$$

$$\mathrm{x} = b_4 \cdot b_5 \cdot z[Ln(b_5 \cdot z) \cdot b_6] = 2.370 \cdot 1.080 \cdot z[Ln(01.482 \cdot z) \cdot 0.938] \qquad (4)$$

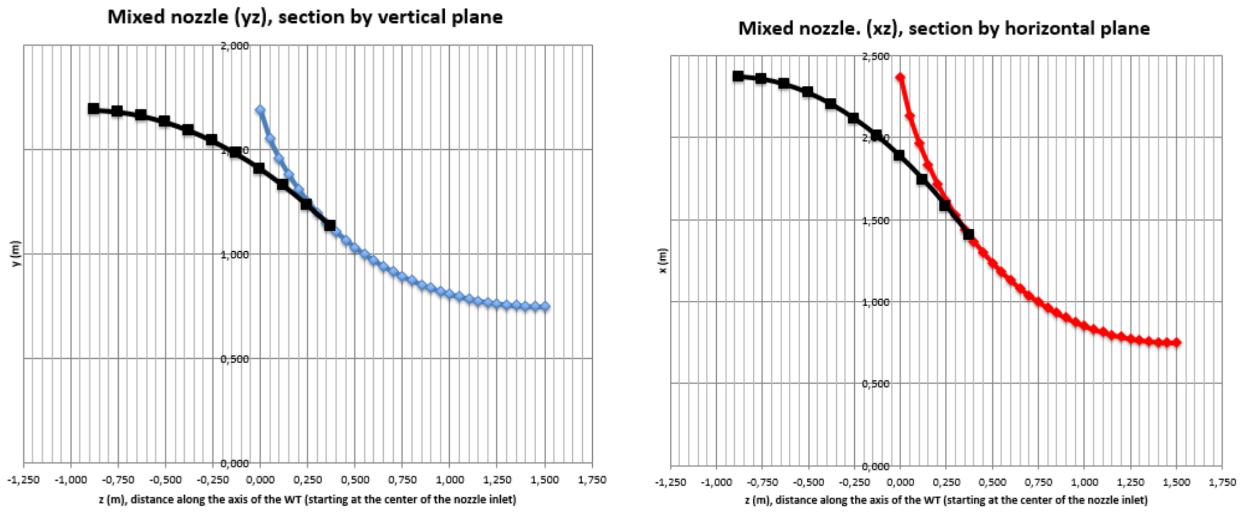


Figure 2. Logarithmic curves (blue and red) for the contraction, both vertical (left) and horizontal (right) superimposed on polynomial third degree curves that are usually used (black).

2.3. Settling chamber and diffuser 2

Once the flow exits the fan (recommended fixed straightener placed downstream), the diffuser drives the flow and the stabilization process starts in the settling chamber. Here the flow increases the pressure by reducing his speed. It is a simple constant section duct, which connects the exit of the diffuser with the entrance of the contraction. The diffuser (a low angle is required) should guarantee a smooth developing of the flow, avoiding the separation of the boundary layer on their walls.

For this diffuser (diffuser 2) it has been decided to design a straight tube after the first curve whose critical parameter is the angle. The length of the diffuser is 7 m and the angle finally obtained is 13°. The value of this angle as well as the existence of the first curve open to possible separations and non-uniform flow distributions, so some type of straightener will have to be installed.

Nevertheless, when a high quality flow is required, some devices can be installed to increase the flow uniformity and to reduce the turbulence level at the entrance of the nozzle. Most commonly used devices are honeycombs and screens. The former reduces drastically the lateral turbulence and does not introduce great pressure loss; screens instead are very efficient in reducing the longitudinal turbulence and imply relatively high losses.

The “honeycomb” is handmade and has an approximate porosity of 83.7%, a cell size of 3.5 cm and a length of 25 cm. Once the first tests are done, at least one screen will be installed.

2.4. Fan

According to our experience, for an open circuit wind tunnel, eventually including settling chamber screens or/and “honeycomb”, the power plant should provide above 100% of the dynamic pressure in the test chamber, apart the efficiency of the motor. This means that:

$$\Delta\mathrm{P} = \frac{1}{2}\cdot\rho\cdot{U_0}^2 = \frac{1}{2}\cdot 1.2\,\frac{kg}{m^3}\cdot\left(50\,\frac{m}{s}\right)^2 = 1{,}500\ Pa \qquad (5)$$

$$\mathrm{Pot} = \mathrm{Q}\cdot\Delta\mathrm{P} = 112.5\ \frac{\mathrm{m}^3}{\mathrm{s}}\cdot 1{,}500\ Pa = 168{,}750\ W \qquad (6)$$

To provide the pressure jump and flow required for the design speed, there were two alternatives: a single fan of approximately 3 m in diameter and with a power of 200 kW; and four fans of 1.3 m in diameter and 45 kW each arranged in a 2x2 matrix (see Figure 3).

Finally, both for economic reasons and the space available, the second option have been installed.

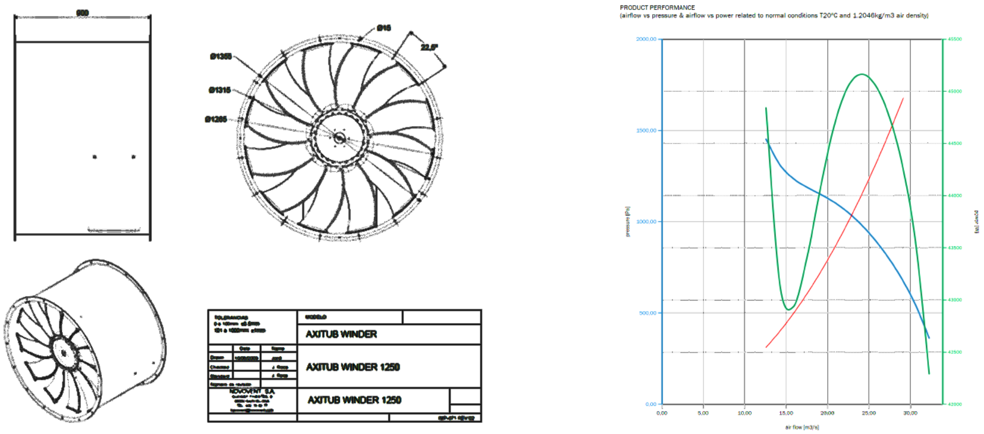


Figure 3. Views (right) and operating curves (left) of the selected fans, AXITUB WINDER 1250.

2.5. Diffuser 1

The last element that closes the circuit is a curved diffuser that connects the end of the test chamber with the section of the fans. The diffuser has been placed inside the curve itself to decrease the speed of the flow as fast as possible and because of the geometric conditions of the place. It will be also necessary to build a straightener as we will see in the next chapter about numerical simulation.

In order to present the project to different Organizations that can collaborate with the financing of the tunnel, a model of the tunnel has been made by 3D printing, making a previous digital design (Figure 4). This model is also used for the teaching, allowing the students a clear and intuitive understanding of the facility.

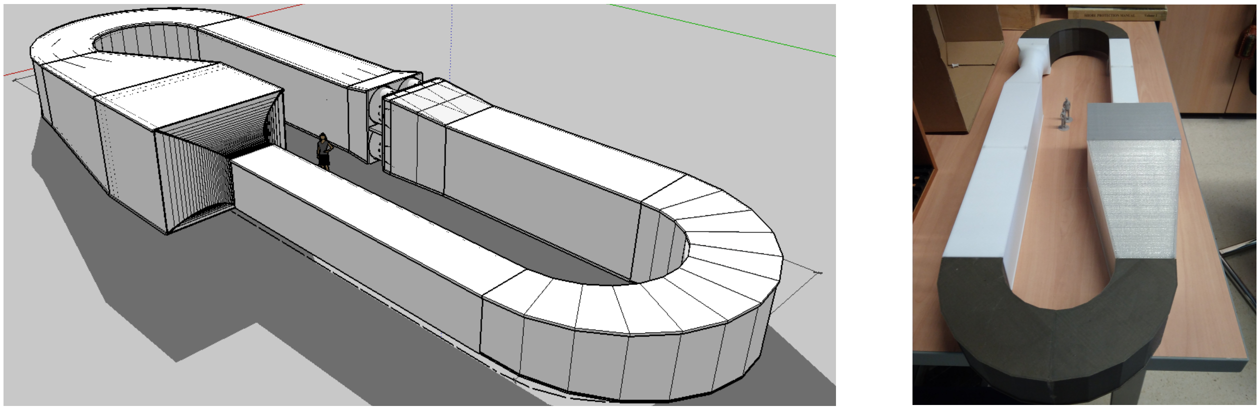


Figure 4. Digital model of the WT of the EPM (left) necessary to make the model printed in 3D (right).

3. Numerical Simulation

3.1. Development of the CFD model

In order to define a more refined model of the wind tunnel constructed a CFD computational mesh is made with software Fluent V16.2. This model must reproduce in the most accurate way the different elements of the wind tunnel described above.

The simulation is performed in steady state and subsonic regime. The tunnel has been modeled at a scale of 1:1. The characteristics of the two models developed are described below:

1. The first one contains a fan in its original position and does not incorporate any type of flow straightener. This model has a number of elements (finite volumes) of 605,674 tetrahedrons; the maximum aspect ratio is 1.58, the maximum "skewness" is 0.68 and the calculation time of each case is about 3.5 min. for 300 iterations in stationary flow.

2. Due to the results previously obtained, a second model is built with the four fans varying their position, incorporating flow straighteners in the curves and simulating the set of "honeycomb" and "screens" as a porous boundary condition. This model has a number of elements (finite volumes) of 1,046,841 tetrahedrons; the maximum aspect ratio is 1.57, the maximum "skewness" is 0.65 and the calculation time of each case is about 5 min. for 300 iterations in stationary flow.

In both cases, the solver SIMPLE is used with a second order spatial discretization of the variables. These parameters allow the resolution of a high number of cases with a reasonable low computational cost.

The atmospheric air is simulated as an incompressible ideal gas which properties are constant and as follows: 298 K, 101,325 Pa, 0% humidity, density ρ = 1.185 [kg m^{-3}], dynamic viscosity = 1.831x10^{-5} [kg·m^{-1}·s^{-1}].

An unstructured mesh has been used, mainly because it guarantees the convergence of the solution in a great majority of initial and boundary conditions for the proposed problem. A sensitivity study of the mesh was carried out, finding that the values of the fundamental variables of the problem are independent of the number of elements of the model from half a million elements. Currently, students are developing several Master's Final Projects in which the possibility of refining the mesh in some areas of the tunnel is analyzed.

The adopted turbulence model has been the k-ε RNG with standard wall functions. The maximum residual errors (RMS) corresponding to the continuity, momentum and k-ε have been limited to 10^{-5}.

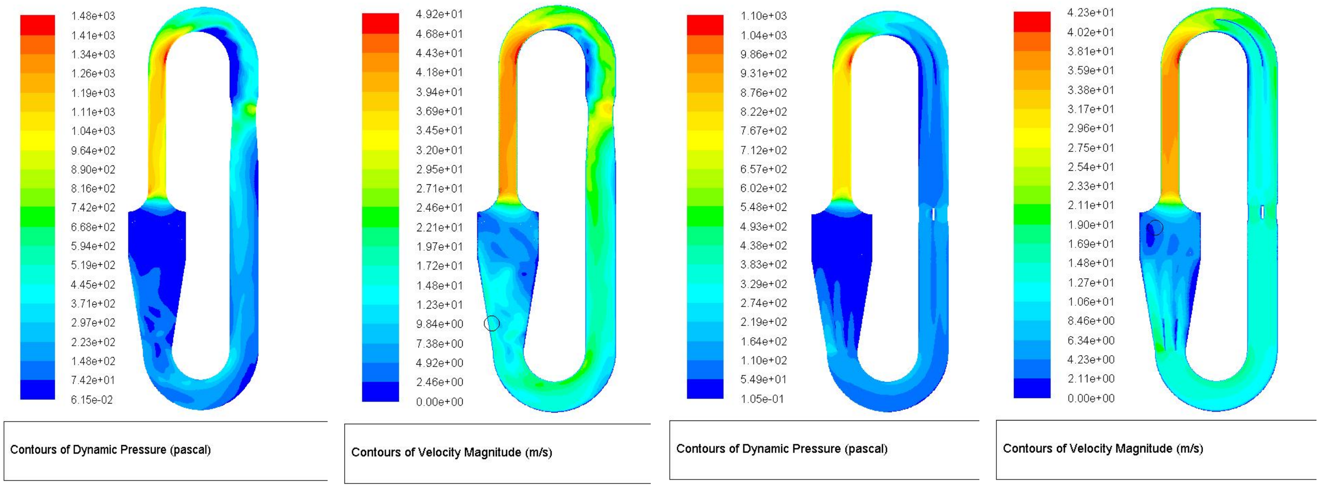
The boundary conditions are the following:

- Fan: it is introduced by approximating the curve of the machine to two straight segments intersecting at the optimum point of operation. Pressure jump at the boundary condition has been set as follows: (26 m/s; 2,000 Pa), (28.64 m/s; 1,000 Pa); (31 m/s; 0 Pa). It is verified that the influence of the mode of introducing this boundary condition has little influence for the analysis that here is intended.
- Walls: no-slip condition is defined. The value of the roughness has little influence because we are in a fully developed turbulent flow.

To facilitate the final convergence of the solution, a standard initialization of the entire domain is performed.

3.2. Results

Figure 5 shows the pressures (Pa) and speeds (m/s) of the two simulations performed. The simulated wind speed in the test chamber is 30 m/s.



(a) (b) (c) (d)

Figure 5. Pressure (a) and speed (b) of model 1 and pressure (c) and speed (d) of model 2; in the middle horizontal plane of the tunnel.

It is observed that in the first model there is a strong detachment in the diffuser 1; diffuser 2 also exhibits a bad behavior. In addition, the speed distribution in the test chamber is not as uniform as it is intended, with so high turbulence values. Therefore, it is decided to modify the position of the fan about 7 m and design flow straighteners in both the variable cross section curve of the diffuser 1 and the diffuser 2. Moreover, the set of "honeycomb" and "screen" is also simulated. With these modifications the behavior of the tunnel, in terms of the described parameters, is greatly improved.

4. Construction

The construction of the wind tunnel is being carried out at the Hydraulic Engineering Area laboratories. All the ducts and necessary elements are being made with DM agglomerated wood and with plywood in some elements, as well as pine strips: DM panels of 2,440 mm x 2,440 mm x 30 mm; plywood panels 15 and 10 mm thick; and strips that have been necessary to cut in order to adapt them to the imposed geometry, trying to optimize their dimensions and to make the construction easier.

Construction was initiated in November of 2017 with the setting out of the wind tunnel on the floor of the laboratory, continued with the construction of the wooden basement that supports all the sections (Figure 6). After that, the walls and ceilings of the different segments were being erected, being necessary in many cases to wedge the base to obtain the perfect union between the different modules that form it.

Currently, with almost all the finished tubes (test chamber, settling chamber, nozzle, diffusers and return duct), the manufacture of internal elements that improve the quality of the flow is being carried out ("honeycomb", "screens" and flow straighteners) waiting for commissioning of the power generator system in July 2018.

Figure 6. Construction of the wind tunnel in the EPM; construction of the basement on the left, the test section and the nozzle on the right.

5. Conclusions

The process of design and construction of wind tunnel carried out by the students and professors of the University of Oviedo, in the Polytechnic School of Mieres, has been described in this paper.

From the point of view of the student, this project supposes the opportunity to understand the need of studying the influence of wind on civil structures, and to develop some research in this field of fluid mechanics. At the same time, the construction of the wind tunnel allows to promote precise learning about this type of research tool. This has been an innovative and enriching aspect for the students themselves, who receive a practical complement to the theoretical knowledge taught in the Master.

This wind tunnel will be used for teaching purposes and for research projects. When used for research, the choice of appropriate measuring equipment (force balance, Pitot tube, hot wire anemometer and pressure scanner) is fundamental to obtain reliable and accurate measurements. In the case of the so-called Civil Aerodynamics or even in educational applications of any field, the requirements related to the quality of the flow can be more permissive, in comparison with the aeronautical ones. Although permissive does not mean ignored.

The main specifications for a wind tunnel are the dimensions of the test section and the maximum desired operating speed. Along with this, the quality of the flow, in terms of turbulence level and flow uniformity, should be specified according to the applications. The quality of the flow, which is one of the main characteristics, is the result of the entire final design, and can only be verified during the calibration tests, which will be carried out this summer. However, according to previous empirical knowledge, some rules can be followed to select the appropriate values of the variables that affect the associated quality parameters.

The aerodynamic forces (drag, lift and moment coefficients, as well as their oscillations), the distribution of pressure along the faces of the object and the velocity field around it can be determined with accuracy, taking into account the laws of similarity: geometric, kinematic and dynamic. They can also be analyzed some aeroelastic phenomena such as vortex-induced vibration (VIV, "Vortex Induced Vibration").

Since the design of the wind tunnels depends mainly on its purpose and therefore, two types of velocity profiles can be reproduced: the first, the uniform profile of which we have spoken; the second, in engineering and architectural constructions and because the buildings are placed on the ground and generally have a relatively low height, they are inside the atmospheric boundary layer (ABL, "Atmospheric Boundary Layer"), it must be implemented a longer and closed test chamber to simulate the equivalent boundary layer, in terms of average speed, profile shape and intensity and turbulence scales. This last case is a more challenging problem, and it is the one that has been addressed in this construction.

6. Acknowledgments

The authors thank the collaboration of “Media Madera Ingenieros Consultores”, specially to Juan Carlos Santos and Julio Vivas; Juan Carlos Mariño, Iván Mariño and Nieves Castrillón, at Doc&Doc Construction Company; and all the students at University of Oviedo collaborating in this project. Besides we would also like to thank Swanson Analysis Inc. for the use of ANSYS University program and Workbench simulation environment. Finally, we would like to acknowledge the help of IDEPA, Principality of Asturias.

7. References

“Wind Tunnel Designs and Their Diverse Engineering Applications”, Edited by N. A. Ahmed (Several Contributors). Copyright © 2015 Second Edition, ISBN-10: 953-51-1047-0, ISBN-13: 978-953-51-1047-7.

Navarro-Manso, A., Fuentes Alonso, S., Alonso Martínez, M., Marigorta Blanco, E. III International Conference Learning of Structural Engineering, ACHE. Valencia, Spain (2013). “Enhancing civil aerodynamics learning: solar panel testing carried out in a wind tunnel”. Publication number: ISBN 978-84-89670-77-8.

Navarro-Manso, A., Coz, J.J., Castro, D., López, M.C., Navarro, A. y Simón-Talero, J. V Structures International Conference (ACHE). Barcelona, Spain (2011). “Analysis of a swing Footbridge over Nalón River by means of CFD and DOE”. Publication number: ISSN/ISBN 978-84-89670-73-0, vol 1.

Meseguer, J., Barrero, A. and Alonso, G., “Ensayos Aeroelásticos de Puentes en Túneles Aerodinámicos”, Hormigón y Acero, VOL 61, nº 256, 2010, pp. 71-81.

Davenport, A.G., “Wind Tunnel Testing: a General Outline”, Wind Engineering Group of The University of Western Ontario, Canada, 2007

Barrero Gil, A., Alonso Rodrigo, G., Meseguer Ruiz, J. and Astiz Suárez, M.A. “Ensayos en túnel de viento de un modelo aeroelástico del arco del puente sobre el río Tajo Arcos de Alconétar”. Hormigón y Acero, nº 245 3er Trimestre. ACHE. Madrid. 2007.