JULES VERNE 2.0, RENEWAL OF A LARGE WIND TUNNEL FACILITY

Pierre PALIER, Anthony COUZINET, Jean-Paul BOUCHET, Sylvain AGUINAGA, Olivier FLAMAND, Philippe DELPECH

Centre Scientifique et Technique du Bâtiment, Nantes - France

pierre.palier@cstb.fr

Abstract

The paper aims at showing the evolution of methods to design a large wind tunnel. The current Jules Verne facility was designed with a scale model of the wind tunnel which enabled hot wire local wind speed measurements. The new facility is designed according a numerical modelling approach which parameters were validated by PIV measurements in the reduced scale physical model.

INTRODUCTION

The Jules Verne climatic wind tunnel of CSTB has been operated for more than 25 years by now; with the complementary dynamic and thermal circuits the facility operates in various domains of expertise, building, civil engineering, energy production, transport… The renewal of the Jules Verne climatic wind tunnel is essential today to maintain its position amongst the major experimental facility at international level.

The modernization of the wind tunnel aims at meeting new societal challenges - energy and ecological transition, safe, clean and efficient energy production, health and wellness, smart, green and integrated transport, climate change ... - and contribute to the development of new induced markets.

These new challenges require the establishment and development of complementary scientific and technical competences related to the expected innovations along with accompanying training actions. On these issues, is added the development of digital technologies, data management and communication technologies, whether for simulations aspects of physical phenomena, as those related to building information modeling (BIM), GIS and Smart Grid.

The Jules Verne wind tunnel modernization project includes the following investments:

- Optimization and adaptation of the geometry of the dynamic wind tunnel unit to optimize the aerodynamic field in order to address specific markets,
- Adaptations of the climatic wind tunnel to meet the needs of key customers: variable geometry nozzles for testing large civil engineering structures (bridges), wind field over urban district; 4WD bench for 4x4, hybrid or full electric vehicles; load bench for off-road vehicles; moving belt floor for the railway industry, dust control systems, etc.
- Regarding the health issue and operating safety of our facilities, it is also planned to replace the cooling tower refrigeration units (3 MW) by dry condensers,
- Expansion of the control rooms, preparation halls, and availability of offices and meeting rooms to welcome outside researchers and corporate partners,
- Digital simulation capabilities to complete the testing service offers with virtual simulations.

THE JULES VERNE CLIMATIC WIND TUNNEL

The Jules Verne climatic wind tunnel is composed of two aerodynamic units: the external circuit corresponding to the dynamic loop (SC1) which can be closed or opened following the kind of applications and the internal circuit corresponding to the thermal loop (SC2).
These two wind tunnels are complementary to the test conditions they allow to reproduce and have independent propulsion systems: a single fan of 6.2 m diameter in SC2 and 6 fans of 3.2 m diameter in SC1. The various test rigs available afford to vary the operating conditions according to the technical capacities and the geometries of the structures tested: wind speed, ambient temperature, icing precipitation, solar illumination...

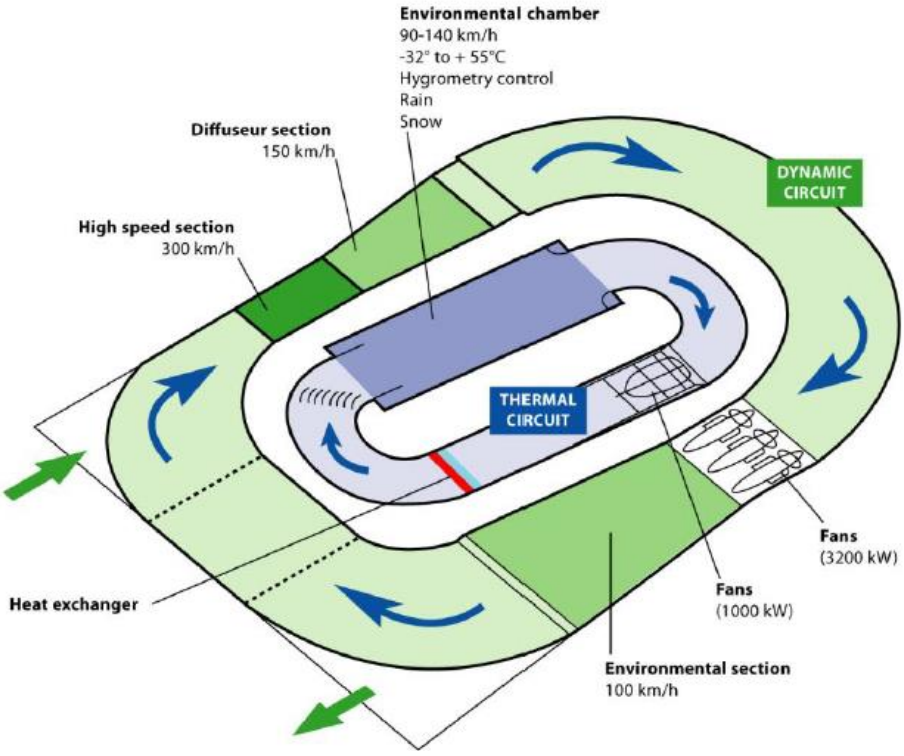


Figure 1: The Jules climatic wind tunnel composed of two units: dynamic and thermal

The specific characteristics of the system offer large use to answer many different subjects (Figure 2).

Figure 2: Highly versatile use of the Jules climatic wind tunnel

Nevertheless, the evolution of customers' requirements and opening up to potential new markets has motivated revisions and improvements of some features of the dynamic wind tunnel. The start of the work is planned for August 2017 and will end in 2018.

DYNAMIC LOOP IMPROVEMENTS OF THE WIND TUNNEL

The main modifications relate to the dynamic circuit of the installation. Indeed specific applications require to have a larger test rig. The geometrical changes have to integrate modularity in order to make it possible to arrange alternately a wide and low or high and narrow test rig.

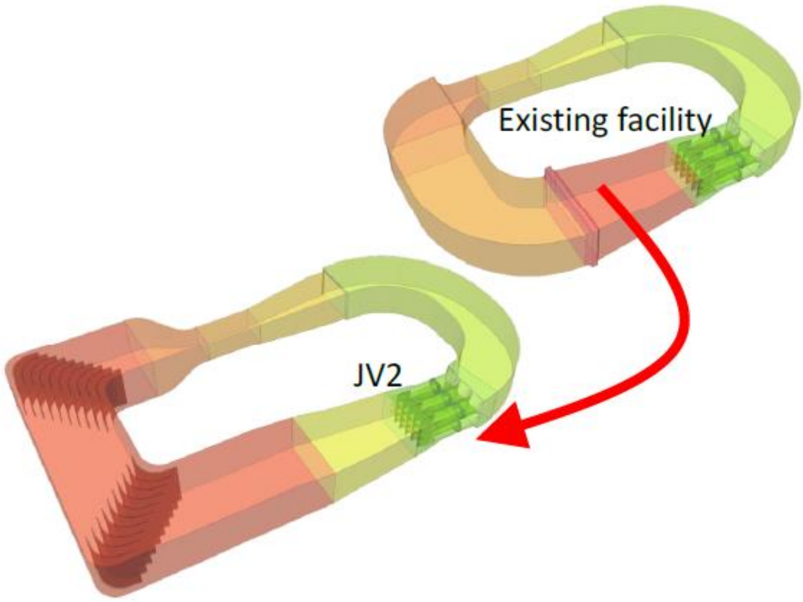


Figure 3: Modification of the Jules Verne wind tunnel geometry

So, a modification of the loop between the environment section and the high-speed section (figure 1) has been proposed; the final proposal is to lengthen the existing facility by replacing the elbow at 180 ° and the asymmetric nozzle contraction upstream of the high-speed vein by two 90 ° elbows containing a series of 9 or 10 vanes in each elbow depending on the width of the test section. This modification has several advantages which have been validated both numerically and experimentally:
Uniformity of flow field is improved in the high-speed test section, tangential velocity components is drastically reduced and the separation flow in the environment test section is limited.

Thanks to optimization works of the flow conditions in the environment test section, new markets could be addressed by studying of large bridges, or ventilation at the scale of a district, or alternately on slender elements with a small scale like pylons, towers or wind turbines. In the high-speed test section, the modifications will allow to improve the accuracy of measurements on reduced scales (1/7th to 1/15th scale) for land vehicles applications, in particular rail vehicles.

Other modifications to the thermal circuit will be made during the modification of the installation: equipment of engine load benches to dissipate power on a brake, installation of a 4x4 test bench operational in the context of severe climatic environments reproduced in the wind tunnels, etc.

METHODOLOGY TO DEFINE THE NEW GEOMETRY

Analytical approach

Definition of guide vanes

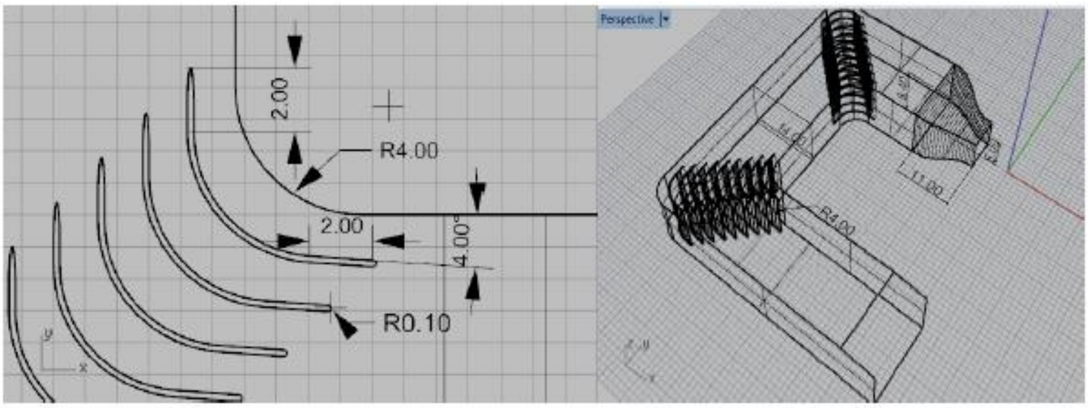


Figure 4: Definition of geometrical elements – guide vanes and contraction nozzle

Contraction nozzle design

Based on Bell et Metha (1987)

$$h(\xi) = (-10\xi^3 + 15\xi^4 - 6\xi^5)(H_{inlet} - H_{outlet}) + H_{inlet}$$

Experimental investigations

The design and validation of the geometrical transformations of the aeraulic circuit of the installation made an intensive use of the CFD, as well as on PIV measurements on a scaled model (1/36) of the future facility. The scale of the model was chosen to allow the use of small fans adapted to the simulation of the air flow and constraints related to the implementation of the PIV measures.

Figure 5: Model of the future wind tunnel made of transparent material in order to enable PIV measurements

Numerical simulations

The computational domain of the wind tunnel is based on the CAD files of the wind tunnel at scale 1. ANSA software has been used to generate the computational grid. It lies on an unstructured mesh which

is composed of tetrahedral volumic elements and prismatic elements layers in order to describe accurately the boundary layers (figure 6). Typically this kind of meshes contains approximately 3.5 million of cells.
Numerical simulations have been performed using open source software OpenFOAM.

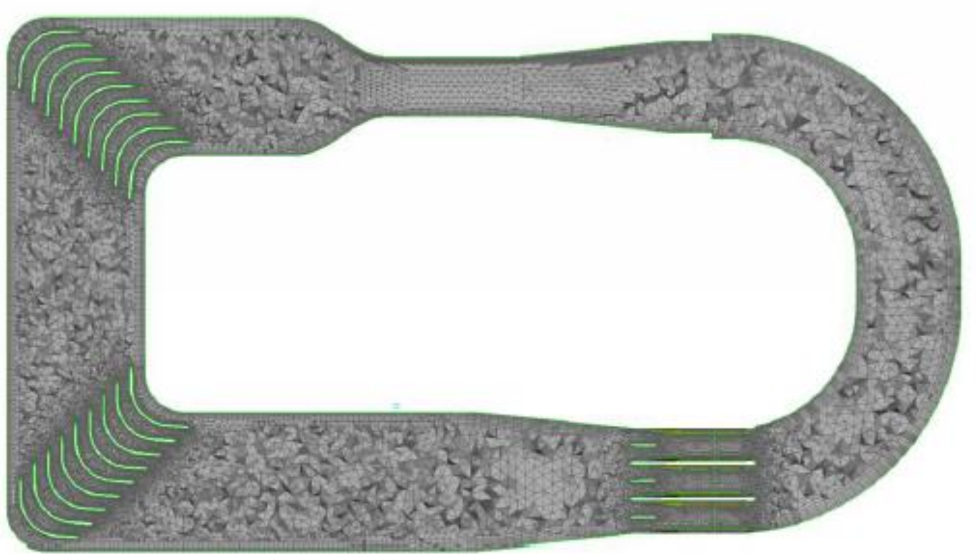


Figure 6: unstructured volumetric mesh based on tetrahedral elements with prismatic boundary layer at the walls

The six fans of the wind tunnel are not taken into account in the computational domain. Six equivalent surfaces have been defined and a constant velocity is imposed on each patch.
Then the boundary conditions are defined assuming:

- The same velocity imposed for each patch without taking into account tangential components
  All the simulations have been simulated with a velocity corresponding to 45 m/s in the high-speed test section;
- Constant pressure is imposed downstream of the fans.

RANS modelling approach is used to simulate the flow field in the wind tunnel. The choice of the turbulence model is based on comparisons with PIV measurements realized on the scaled mock-up. The comparison of the separation flow induced by the interaction between the diffuser (in diffuser test section cf. figure 1) and the elbow at 180° downstream of the fans proves to be a good parameter to evaluate the turbulence models.

Figure 7 compares results from PIV measurement, standard k-ε modeling and k-ε realizable modeling. The latter seems to reproduce the detachment flow in the diffuser more faithfully. Indeed, this kind of model are driven by a viscosity limiter function which tends to cut the turbulent viscosity in zones of adverse gradient inducing the detachment effects and making it more adapted to this flow configurations.

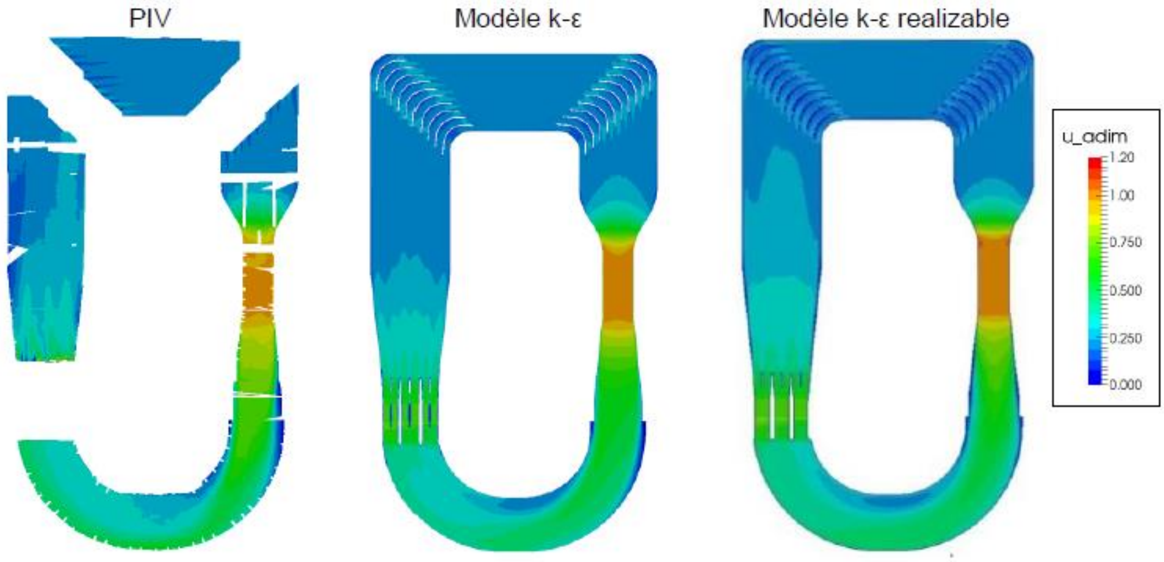


Figure 7: Comparison of turbulence models to PIV measurements

ADAPTATION TO REQUIREMENTS OF THE ENVIRONMENTAL TEST SECTION

As shown in Figure 8, the modifications of the wind tunnel offer a larger environmental test section. Moreover the velocity field is sufficiently uniform to consider carrying out tests in this zone.

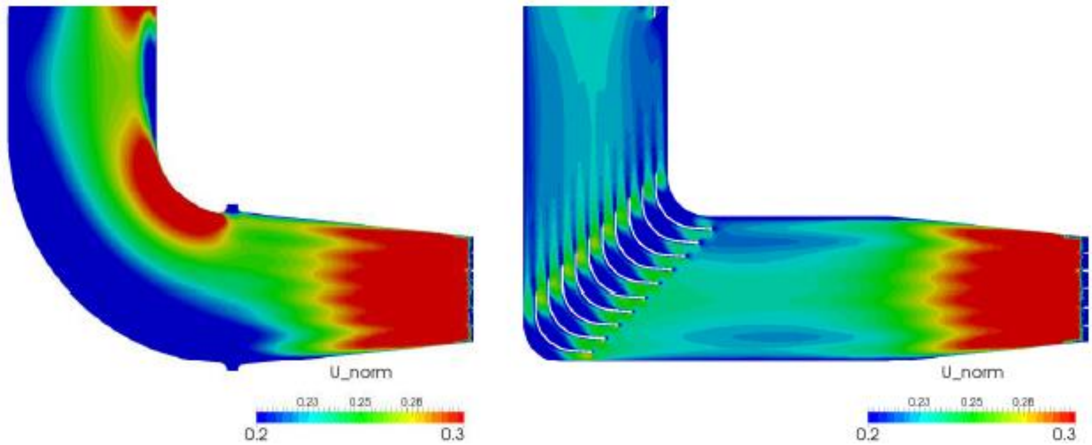


Figure 8: Visualization of the velocity field in the environmental test section – SC1 vs future wind tunnel

The possibility of integrating vertical or horizontal "blowing nozzles" into the environmental test section has been studied numerically and experimentally. The provisions planned for these additional nozzles are (figure 9):

- vertical for the study of pylons, masts, wind turbines ...
- horizontal for studies with aero-elastic decks at large scales, districts, wind farms,

Figure 9: Fitting of vertical (left) or horizontal nozzles (right) in the dynamic unit of the Jules Verne wind tunnel

The expertise carried out using numerical simulations aimed at checking the quality of the flow in the proposed test area and evaluating the average aerodynamic loading applied to these nozzles, the originality of which is based on the implementation of inflatable structures by assuming that these equipments are rigid and undeformable.

Vertical nozzle configuration

The main interest of using a vertical nozzle in the facility is to be able to test slender structures, pylons, mats, wind turbines, etc., in a high-speed wind stream without additional energy constraints. For this purpose, two vertical symmetrical profiles would be placed in the test section, allowing a cross section of about 50 m² (6 x 8.40 m).

The results of the CFD calculations of the velocity fields at the exit of the vertical nozzle show the existence of a bifurcation of the wind field towards the outside of the turn (Figure 10), which is homogeneous over the entire height of the section. The 9 inter-blade channels are then fed heterogeneously (supercharging channels 2-3-4-5). The shear zone is marked significantly from the nozzle outlet. It should be noted that this average representation should not hide the important unsteady effects in this zone which may impact the central zone of the jet. The qualitative comparison of the results obtained by numerical simulations and PIV on the 1 / 36th scale model (Figure 10) confirms this result, although numerical simulations tend to predict a more spread velocity gradient in the shear zone.

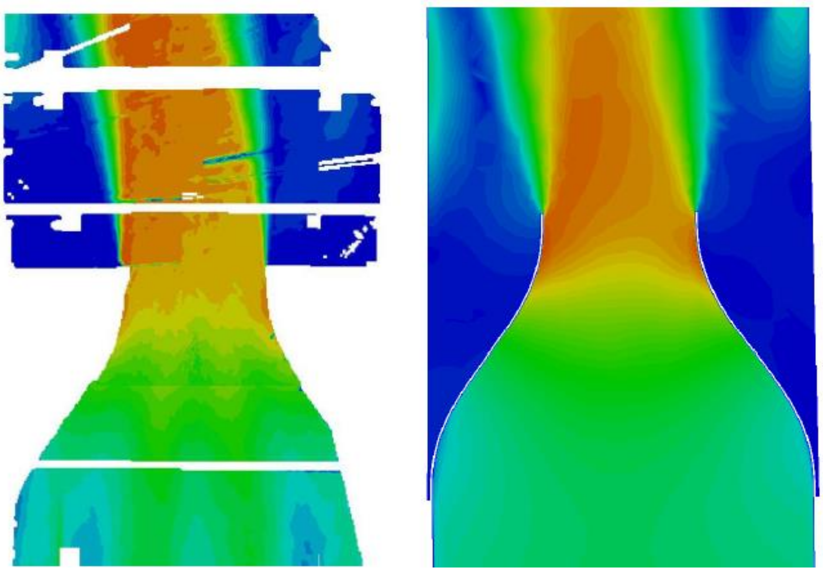
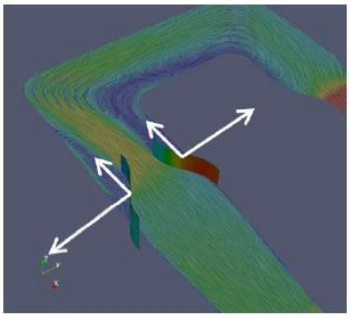


Figure 10: Comparison of the wind velocity field downward the wind nozzle by PIV (left) and CFD (right)

From an aerodynamic point of view, it seems that this solution does not guarantee a clean flow at the nozzle outlet.

Aerodynamic loading of vertical nozzles

The parietal pressure fields are shown in figure 11.
The bifurcation identified in the preceding paragraph induces a dissymmetry of the pressure field on each of the vertical nozzles (Figure 11). This results in a significantly different average force between the "inner" nozzle and the "outside" nozzle. The maximum effort recorded corresponds to an average load of 22 kg / m².



	Internal wall	External wall
Fx	-6.9 kN	-9.8 kN
Fy	15.8 kN	21.8 kN
F	17.2 kN	23.9 kN

Figure 11: Evaluation and graphic representation of wind efforts applied to vertical nozzles

The bifurcation is due to an imbalance of the pressure in the series of blading. In order to rebalance this pressure, one solution could be to close one or more inter-blade channels on the upper surface of the turn so as to straighten the flow at the nozzle outlet.
Simply in a first step, we propose to close the three inter-blade channels at the outside of the turn as shown in figure 12.

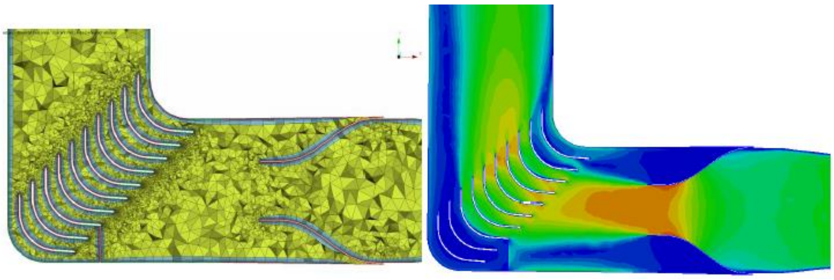


Figure 12: Numerical test of blocking of the guide vanes channels

The implementation of such an adaptation could be easy by using a door system which could be open or closed depending on the test conditions. This first test was carried out numerically to estimate the potential of this adaptation. The effect is straightforward, but the operation seems tricky because the jet tends to bifurcate inwardly in this configuration. This bifurcation having been identified on the model, it is then proposed to study two configurations by closing two or three inter-blade channels as shown in figure 13.

Figure 13 compares two experimental solutions which aim at closing more or fewer inter-blade channels in order to check if the evolution of the jet at the nozzle outlet can be controlled. PIV measurements were carried out on the scale model to assess the effect of these changes on the mean velocity field. The PIV visualizations are complex to carry out in this area of the section because the reflections are numerous in the region of the turn. However, it is observed that a simple solution based on the closing the first two channels at the outer side of the turn and the first channel inside the turn makes it possible to obtain a symmetrical velocity field with respect to the axis of the nozzle.

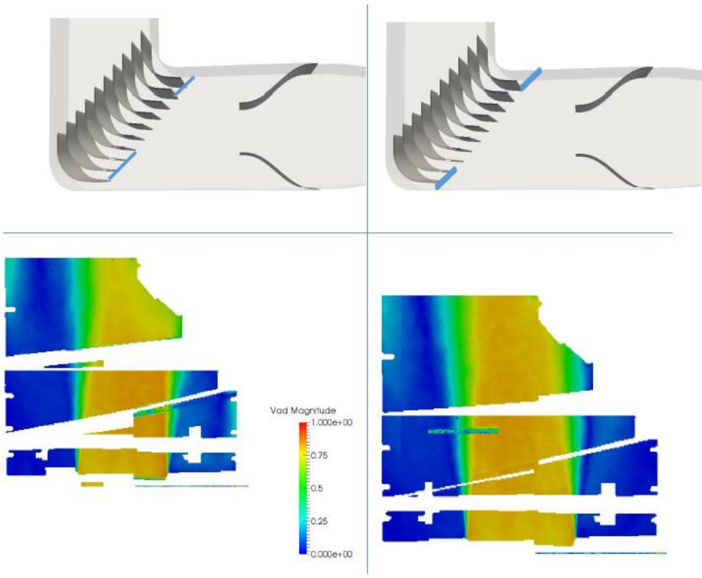


Figure 13: Experimental test of blocking of the guide vanes channels

Horizontal nozzle configuration

The purpose of this configuration is to create a test section that makes use of the maximum width of the test section by lowering the ceiling height from 8m to 3m. The section is then equal to 3 X 15 meters (45 m2). This test section would be created from a single horizontal nozzle which can be placed at a distance from the downstream turn and the outlet of the fans. The contraction generated by the nozzle will reduce the heterogeneity of the velocity field that remains at the fan outlet.
A cross-section of the environment section in the middle of the nozzle makes possible to identify the structure of the velocity field which naturally presents a sheared zone at the nozzle outlet (Figure 14).

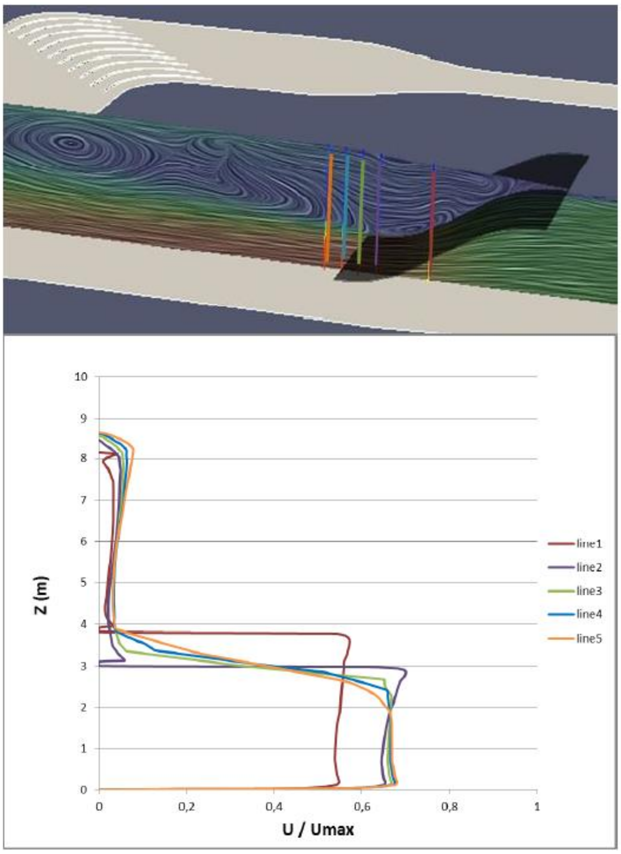


Figure 14: Wind velocity profiles in a vertical plane, upstream the nozzle and in the test section. Evolution of the wind velocity with height at five locations in air flow: in the nozzle, near the trailing edge and three other locations near the nozzle outlet

One can observe an overspeed at the trailing edge generating a low pressure zone. The velocity at the nozzle outlet is of the order of 0.68 times the maximum speed obtained in the wind tunnel circuit. It can be seen that the sheared zone impacts the testing zone, these observations are confirmed by the velocity mappings at different heights of the test section. Again, this result is only a description of the mean field, the unsteady phenomena due to turbulence may impact the test zone. The size of the turbulent structures developing in the sheared zone can be of the order of magnitude of the height of the nozzle. Nevertheless, this field remains two-dimensional as shown by the different velocity fields obtained in the depth of the test section.

It could be envisaged to extend the trailing edge of the nozzle by a horizontal part which makes possible to protect the test zone from the sheared zone.

Aerodynamic loading of the horizontal nozzle

It is possible to represent the pressure field applied to the nozzle with a pressure coefficient based on a pressure at the surface and a reference speed (1).

$$C_p = \frac{P - P_{ref}}{0.5\rho U_{ref}^2} \quad (1)$$

The change in the pressure coefficient as a function of the height of the nozzle has a conventional pressure distribution (Figure 16). The pressure coefficient is maximum at the leading edge. The low pressure zone is located at the trailing edge of the nozzle (between 3 and 3,4 m). If the blowing nozzle is considered to be formed by a thin wall, the low pressure face of this wall has a constant distribution of the pressure field. This obviously has no physical reality if the inflatable nozzle fills all of the space at the rear of the nozzle.

The integration of the pressure field on the nozzle makes possible to obtain the components of the mean force applied to the nozzle. The effort recorded corresponds to an average load equivalent to that obtained on the vertical nozzle for the previous configuration, namely 26 kg / m².

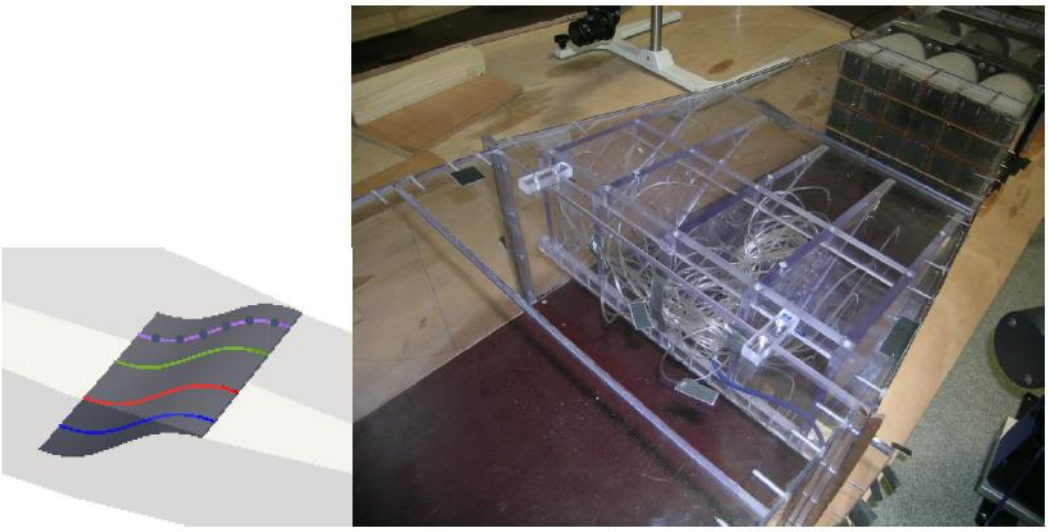


Figure 15: Locating the pressure taps on the horizontal nozzle

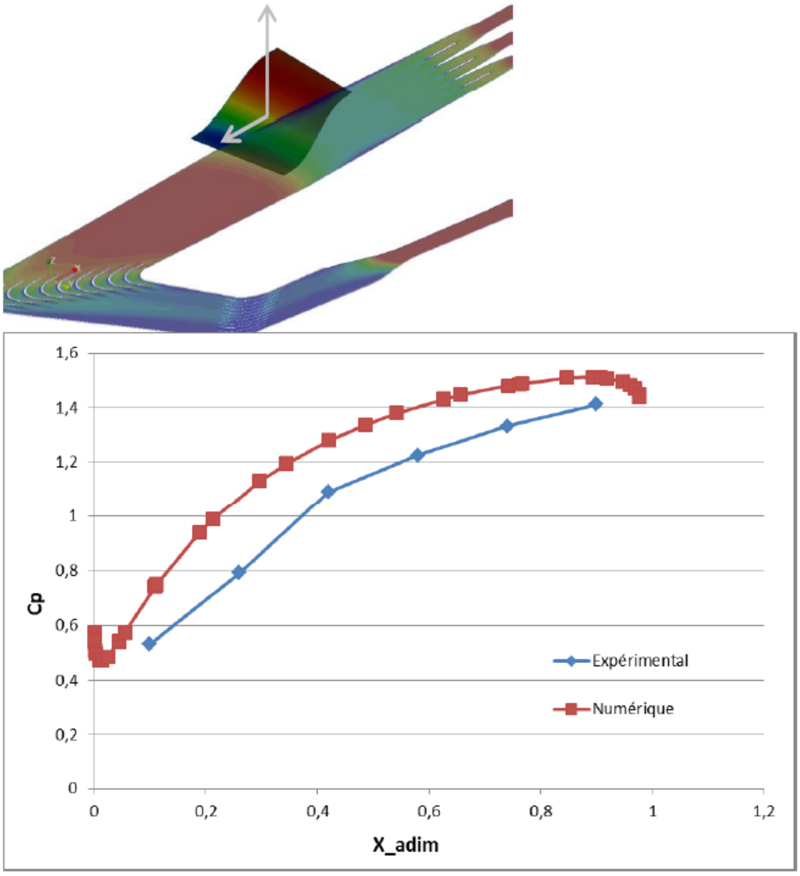


Figure 16: Mapping of the pressure field at the wall of the horizontal nozzle and efforts applied to the nozzle

CONCLUSION

The results presented in the present study allowed us to obtain a great confidence in the both the numerical model and experimental model at reduced scale. As a consequence it is expected that the future configuration of the Jules Verne Wind Tunnel will meet the same specifications as the ones obtained through the monetization.

Additional measurements carried out in the actual Jules Verne wind tunnel are also available abut not presented here. They were used in order to understand the flow behavior downstream the fans. This section of the wind tunnel will be indeed used in addition with inflatable nozzles at it will presented in another paper.