DESIGN, ASSEMBLING AND VERIFICATION OF A CIRCULATING WATER CHANNEL FACILITY FOR FLUID DYNAMICS EXPERIMENTS

Gustavo Roque da Silva Ássi
g.assi@usp.br

Julio Romano Meneghini
jmeneg@usp.br

José Augusto Penteado Aranha
japaran@usp.br

Walter George P. Coleto
walter.coleto@poli.usp.br

All authors from:
NDF – University of São Paulo
Av. Prof Mello Moraes, 2231
Cidade Universitária – São Paulo – SP
05508-900 – Brazil

Abstract. *This paper presents the design, assembling and verification process of the circulating water channel employed for experiments in fluid dynamics at the NDF – "Núcleo de Dinâmica e Fluidos" – "Escola Politécnica" at the University of São Paulo. A brief review compares some possible solutions found in water facilities. The paper justifies the needs of a water channel in Brazil and its potential applications in the whole field of engineering: automobilist, aeronautics, ship science, naval, ocean, offshore, oil and gas, among others. It also describes the design procedures applied to find the geometry of some elements of the channel, especially the contraction, the sink and the propulsion system. Some results using a 1:4 and 1:5 scaled models are presented to predict a few characteristics of the prototype. In addition, the paper presents a list of installed instrumentation, such as: hot-film anemometry, particle image velocimetry, flow visualization and six-force component balance. Some test section results are presented as verification parameters for velocity profile and turbulence intensity employing hot-film anemometry. For two analysed flow conditions, a 10% deviation was found in the cross section velocity profile. Vertical centre line velocity profiles are also shown for four conditions of mean velocity: (0,1; 0,2; 0,3; 0,4)m/s. Besides that, a mean turbulence intensity level of 0,02 was obtained for a wide range of operational conditions. Finally, the paper concludes suggesting some new investigations to improve flow quality.*

Keywords*: water channel facility, fluid dynamics, experiments, hydrodynamics.*

1 Introduction

The NDF – "Núcleo de Dinâmica e Fluidos" (Fluids and Dynamics Research Group) – at the University of São Paulo develops research projects within fluid and structural dynamics, bluff body flow, and flow-induced vibration. At the moment, a brand new water channel facility is being employed to carry some experimental studies. This facility, carefully designed and instrumented, represents an important device to improve detailed experimental investigations. Differently from a towing tank, this circulating water channel provides a precisely controlled test section for an indefinite period of time, as long as it needs to be. Preserving test section conditions trough long experiments allows a better examination and data quality to be acquired.

An experimental channel or tunnel is designed to generate an adequate flow condition to run experiments in a determined test region, called the test section. Generally, a constant and uniform flow velocity profile is required, but some experiments involve different test section conditions. For instance, an atmospheric wind tunnel requires a parabolic velocity profile in order to simulate Earth's atmospheric behaviour. In fact, each experiment will need a specific test section characterization, including: working fluid (liquid or gas); velocity profile, uniformity, turbulence intensity; and so on. In addition, these experimental facilities can also be classified by closed-circuit or open-circuit tunnels. Since water was chosen as the working fluid, a closed-circuit channel would be the natural option. This solution provides good control for flow visualization methods and flow seeding applied to velocimetry techniques. Some experimental tunnels are illustrated at Figure 1 presenting their parts and circuit types.

From now on, this paper will be focused on describing the NDF Circulating Water Channel facility. Complementary information about wind tunnels, water channels and other experimental facilities can be found in recommended literature: Barlow et al. (1999); Gordon & Imbabi (1998); Mehta & Bradshaw (1979); SNAJ (1986).

The NDF Circulating Water Channel is part of the Polytechnique School of the University of São Paulo and became not only an important research facility, but also an educational instrument. For this reason, a versatile design was expected to satisfy all its academic and technological applications. In the next sections a brief report of the design and construction process will be presented, followed by an experimental verification procedure of the test section.

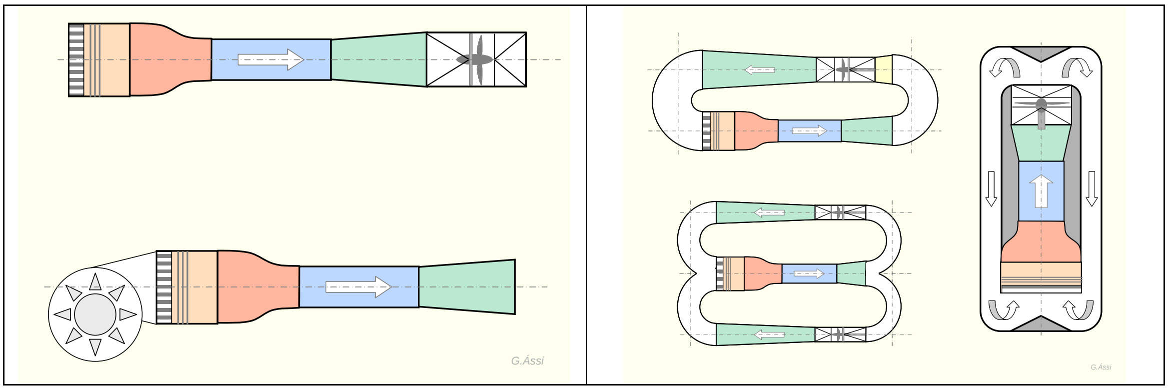

Figure 1: Some examples of (a) open-circuit and (b) closed-circuit tunnels.

2 Conception and Design Process

For about ten years ago, a group of researchers related to offshore risers dynamics (deep sea oil exploration subject) came up with the idea to construct a water channel in order to experimentally verify and evaluate numerical and analytical investigations. By the year of 2003, this target started to come through with the organization of NDF research group, financially supported by Finep, Fapesp, Petrobras and CNPq. A few researches were involved discussing the better solution would satisfy the main needs.

One of the previous conceptions comprised a horizontal closed-circuit water channel with a free-surface *800x600x3000mm* test section, where models and instrumentation can be observed by glass walls. A conventional centrifugal pump would power the flow until $0{,}72m^3/s$, resulting on *1.5m/s* mean velocity profile. In order to verify the flow conditions and free surface behaviour in the test section, a *1:4* Froude scaled model was constructed. Although this solution could reach a higher velocity, because of its circular flow circuit, some limitations on the test section made it to be neglected. This first conception showed to be a workable water tunnel solution presenting a very stable low-turbulence flow quality. At the moment, the scaled model is still employed as didactic facility for graduation classes. Figure 2 shows the *1:4* scaled model and a schematic representation of the first channel solution. Consequently, the more versatile design figured as the better solution, which is detailed in the following sections.

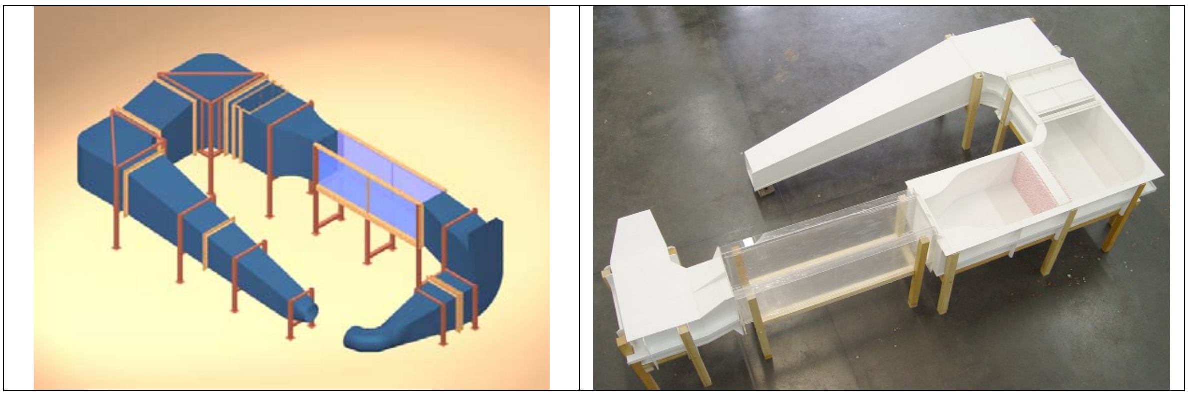

Figure 2: (a) First solution schematic representation and (b) *1:4* scaled model of first conception.

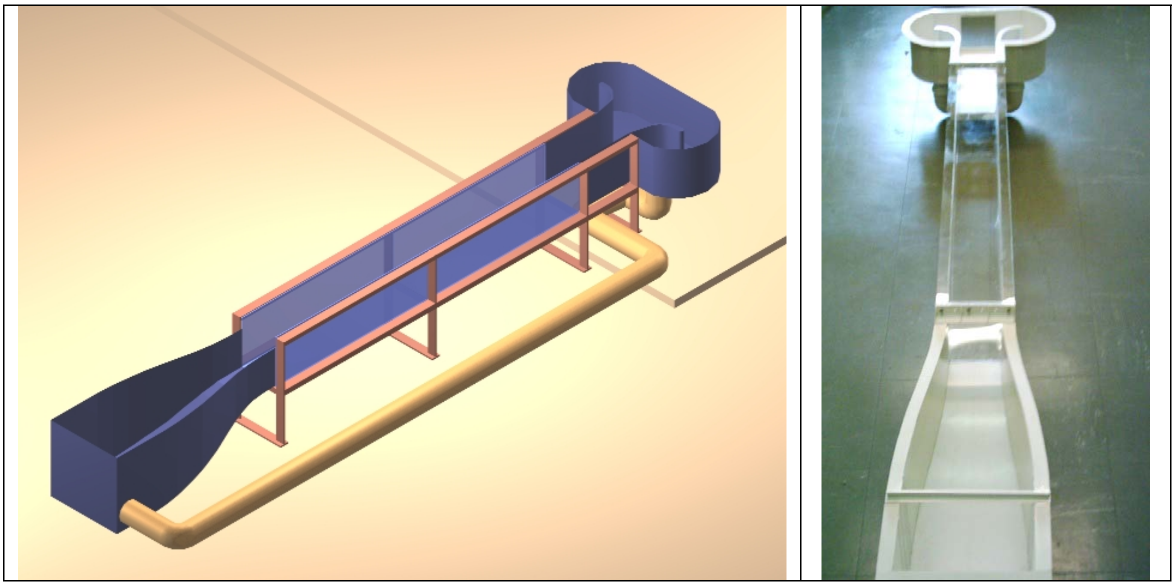

Figure 3: (a) NDF channel schematic representation and (b) *1:5* scaled model of final conception.

3 NDF Circulating Water Channel Specifications

The new conception, called the more versatile solution, presents a working zone capable to carry out a wider range of different nature experiments. With an extensive longer free-surface test section, this channel can be employed either as a water tunnel or a towing tank. In this way, it can be a more useful facility for naval, aeronautics and automobilistic experiments. The principle implementation was based on the Imperial College of London water channel facility, which has its flow circuit closed with circular tubes, providing a better utilization of laboratory area. As consequence, some new elements had to be developed, such as the source-tube inside the contraction module and the curved diffuser within the sink. All the elements are presented in the following sections.

In the same way, a *1:5* scaled model was constructed in order to verify the flow quality, free surface behaviour and flow conditions on the new designed elements (contraction and sink). A scheme of the final solution and the scaled model are presented on Figure 3, while an illustrative blueprint of the channel can be seen on Figure 5. Table 1 shows the NDF Circulating Water Channel operational specification. Figure 6, placed at the end of this paper, presents a view of the water channel facility and laboratory.

Table 1: NDF Circulating Water Channel specifications.

Date of establishment	*December 2004*
Type of circuit	*Horizontal, closed circuit*
Test section	*700x900x7500mm, free surface*
Flow rate	$0,005m^3/s \sim 0,5m^3/s$
Towing carrier speed	*0,005m/s ~ 0,5m/s*
Propeller diameter	*380mm, 3 blades*
Driving power	*30KW*
Contraction ratio	*4:1*

3.1 Test section

Due to the fact that experiments will be conducted inside its parallel-glassed walls, the test section is the most important element of an experimental channel. For this reason, all other elements must contribute for a good quality flow generation. An accurate flow properties control is essential for a successful experimentation, for instance: velocity profile, turbulence level, temperature, waves, among other specified for each situation. The flow is generated and modified in the contraction (described as follow) and received by the test section. In normal conditions, it will take from *0,5 diameter* (test section hydraulic diameter) to reach adequate levels of uniformity (Mehta & Bradshaw, 1979).

NDF facility presents a *700x900x7500mm* test section with a top flow rate of *0,5m3/s*. For a typical water level of *0,7m* the flow velocity reaches *1,0m/s*. Comparing to the first channel conception, the new solution doubled its length with a deeper and wider cross section, providing versatile working zone that can also be used as a towing tank. A step motor controlled traverse system is installed above the section structure working as a high-definition positioning system and model mover support as well. The parallel walls are made of *20mm* toughened and laminated glass planes, providing entire section visualization.

3.2 Regularization chamber and Contraction

In the bottom part of the regularization chamber is located the source tube, which distributes the flow to reach a uniform flow rate through chamber width. Some conditioner elements (honeycomb and grids) are placed to regularize flow velocity and turbulence components. While the aluminium honeycomb works reducing velocity cross components, a pair of small meshed grids equalizes velocity profile. Differently from the honeycomb, the grids set implies a huge pressure loss to the flow, reducing higher velocity axial components to an almost plane profile. Channel chamber has a honeycomb with *9mm* diameter by *160mm* length cells and a pair of stainless steel wire grids installed one just after the other.

Downstream the chamber is place the two-dimensional free-surface contraction. According to the vortex stretching phenomenon, the contraction reduces the remaining turbulence scales by accelerating the flow until the test section entrance. Mehta & Bradshaw (1979) affirm that a contraction geometry design is based on a controlled flow generation on its boundaries, so the boundary layer is maintained attached along the contraction walls avoiding flow separation. In addition, the NDF Channel presents a contraction ratio of *4:1*.

3.3 Sink

A special diffuser solution was designed to prepare the flow, coming from the test section, to be drained inside the duct system. The sink consists of a curved diffuser placed inside a reservoir that decelerates the flow, recovering pressure, before it gets inside the duct system. A careful hydrodynamic design prevents intermittent flow separation that could cause undesired pressure variation and consequent vibration. Employing numerical simulations and scaled model analysis, it was possible to find a better geometry configuration for the sink, which is presented on Figure 4(a).

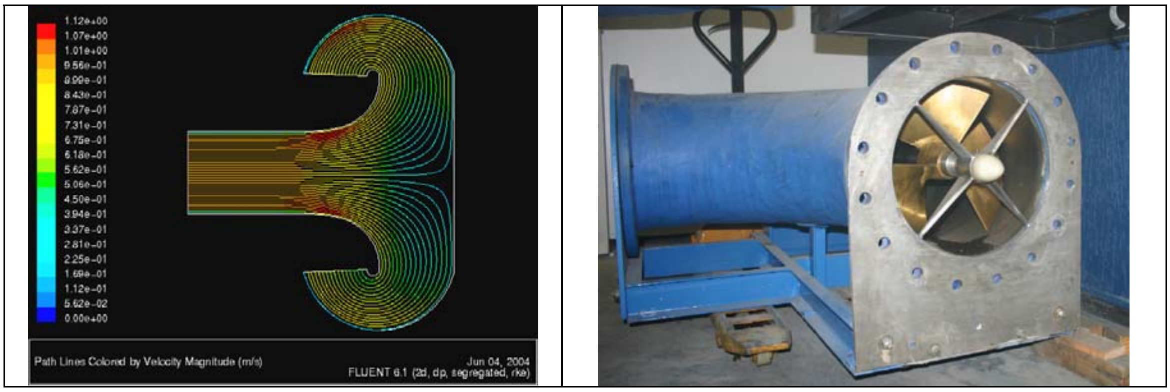

Figure 4: (a) Numerical simulations of two-dimensional flow inside the sink reservoir. (b) Propulsion system, attention to the propeller module before installation.

3.4 Propulsion system

A *380mm* diameter fixed-step impeller produces the necessary power to circulate the flow in the circuit. A shaft connected *30KW* electric motor drives the system up to *900rpm*. With this configuration, a $0{,}5m^3/s$ can be reached. Figure 4(b) shows the impeller unit before its installation.

3.5 Available Instrumentation

The NDF Circulating Water Channel Laboratory is equipped with the following instrumentation set, ready to be employed in every day experiments (Table 2):

Table 2: NDF Circulating Water Channel Laboratory instrumentation

- General purpose data acquisition computer with direct network connection
- National Instruments SCXI system: 8-channel strain-gauge; 8-channel accelerometry; 16-channel multi purpose instrumentation modules. 16-bit analog-to-digital converter.
- Step-motor controlled traverse system used for probe and model positioning or model towing trough channel length.
- High resolution PIV(Particle Image Velocimetry) system (TSI Incorporated): 20Hz pulse laser; 4 mega-pixel digital camera.
- High resolution still camera (Canon 6 mega-pixel); Hi-8 and digital video camera.
- Hot-film CTA (Constant Temperature Anemometry) module (Dantec Dynamics): 8-channel module with temperature probe.
- Flow visualization techniques: ultra-violet lights for fluorescent dye; PLIF – Planar Laser Induced Fluorescence; hydrogen bubbles tracer.
- Six-component dynamometric balance: 3 force and 3 moment components.
- Compressed air line for low-damping air bearings.

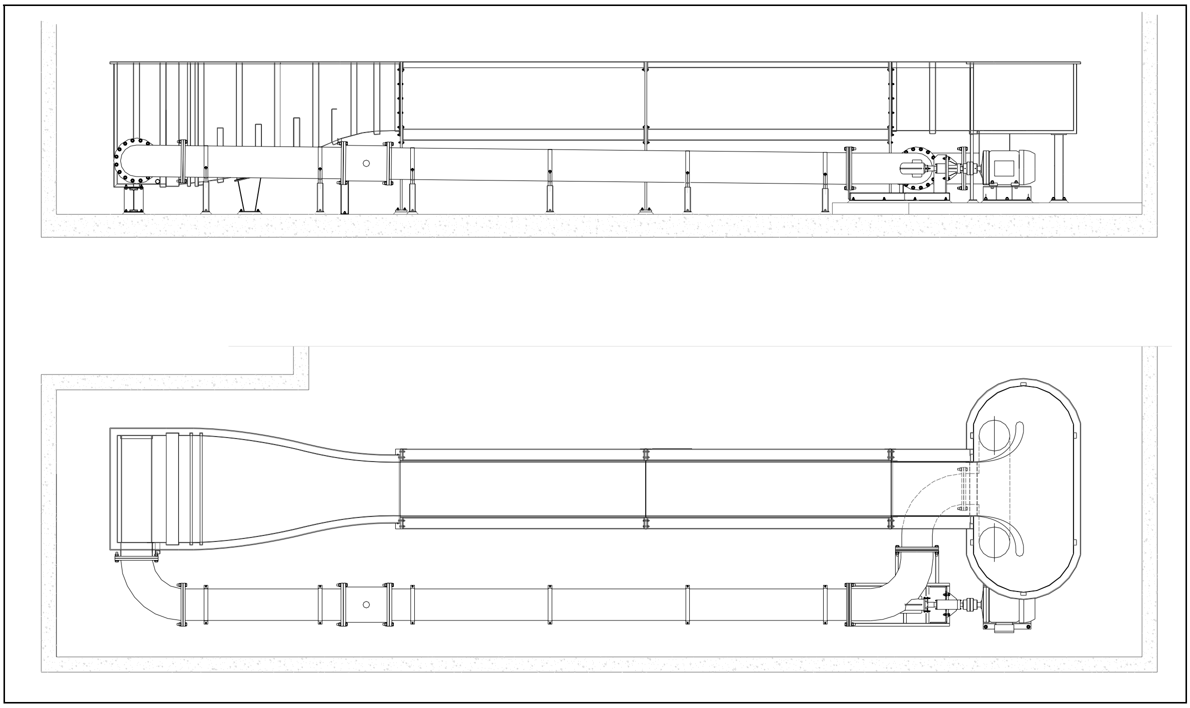

Figure 5: Outline drawing of NDF Circulating Water Channel.

Figure 6: The NDF Circulating Water Channel facility and Laboratory.

4 Experimental results and discussion

4.1 Test section verification

Test section verification represents the first necessary analysis to evaluate the channel quality. This section presents the experimental results accomplished by mapping velocity and turbulence properties with hot-film anemometry technique. The velocity measurements were developed in the first half of the test section, *1,0m* downstream the test section entrance and a hot-film *2mm* long and $2\mu m$ probe was employed. For the interested reader, the literature Bruun (1995) is recommended to revise the hot-wire anemometry, which will not be discussed in this paper.

Considering the time series of total measured velocity ($u(t)$) in terms of a mean velocity ($\bar{u}$) and a fluctuating ($u' = u(t) - \bar{u}$) component, the Turbulence Intensity (*TI*) is defined by the root-mean-square of the fluctuation divided by the mean velocity, as the equation below.

$$TI = \frac{\sqrt{\overline{u'^2}}}{\bar{u}} \tag{1}$$

All length dimensions were nondimensionalized by the test section width (*W=0,7m*). The Reynolds number was calculated employing the channel width, as follow:

$$\mathrm{Re} = \frac{u \cdot W}{\upsilon_{water}}, \tag{2}$$

in which ν_{water} is the water cinematic viscosity.

Vertical centre line velocity profile.

Positioning the velocity probe in the vertical centre line of the test section, the velocity profile is obtained measuring the mean velocity for each position along the line. Figure 7 presents the results for four different flow rates corresponding to *0,1m/s*; *0,2m/s*; *0,3m/s* and *0,4m/s*. The red lines on the graphic represent the standard deviation around the mean velocity through the path.

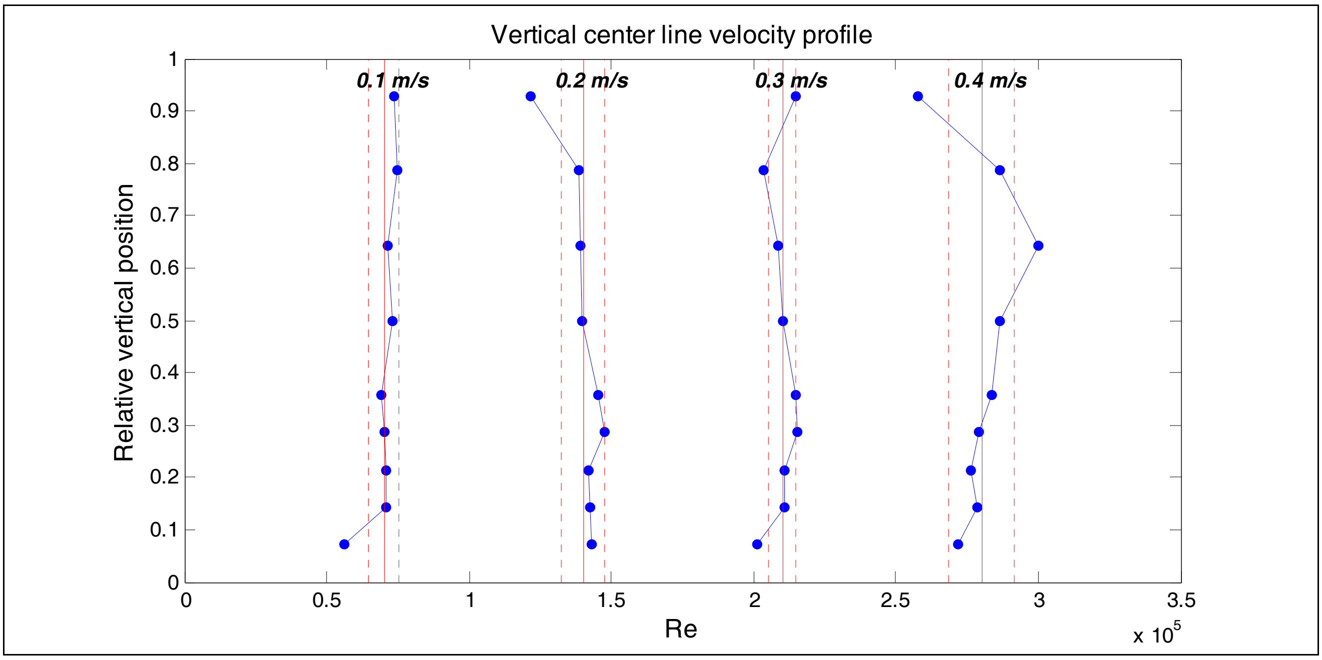

Figure 7: Vertical centre line velocity profile for four flow conditions (*0,1; 0,2; 0,3; 0,4*)*m/s*. The continuous red line shows the mean velocity, while the dotted red lines represent the standard deviation. Vertical position nondimensionalized by test section width (*W=0,7m*).

The first three curves show a narrow variation of *5%* around the mean velocity. An almost plane profile can easily be identified, what is evaluated as a good quality velocity profile. In the other hand, the last curve (*0,4m/s*) shows a higher acceleration closer to the free surface followed by a velocity reduction on the top. This profile behaviour is expected to be found in free surface channels when the velocity tends to increase. In this case, the profile variation around the mean velocity is slightly higher than *5%*.

Although velocity variation close to the free surface will always appear in open channel flows, the authors believe this more than *5%* variation can be minimized adjusting the regularization chamber elements.

Turbulence variation

In order to estimate the turbulence variation with flow velocity, the probe was positioned in the centre point of the cross section. Data was recorded for 14 conditions of flow rate; varying from *0,05m/*s up to *0,5m/s*. Figure 8 presents the turbulence intensity (*TI*) variation with Reynolds number. The continuous red line represents stands for the mean turbulence intensity, while the dotted red lines represent the standard deviation.

Considering this acquired data, the channel presents mean turbulence intensity $TI = 0{,}022 \pm 0{,}004$, what is considered to be a low turbulence level for a water tunnel. However, the *20%* variation can also be minimized improving the grid set in the regularization chamber. A third grid can be added or the mesh density can be increased in order to reduce velocity fluctuations.

In addition, this mean *TI* presented only gives an idea of turbulence intensity order, but cannot be considered the channel reference turbulence index, since it was measured in just one point and not throughout the entire cross section.

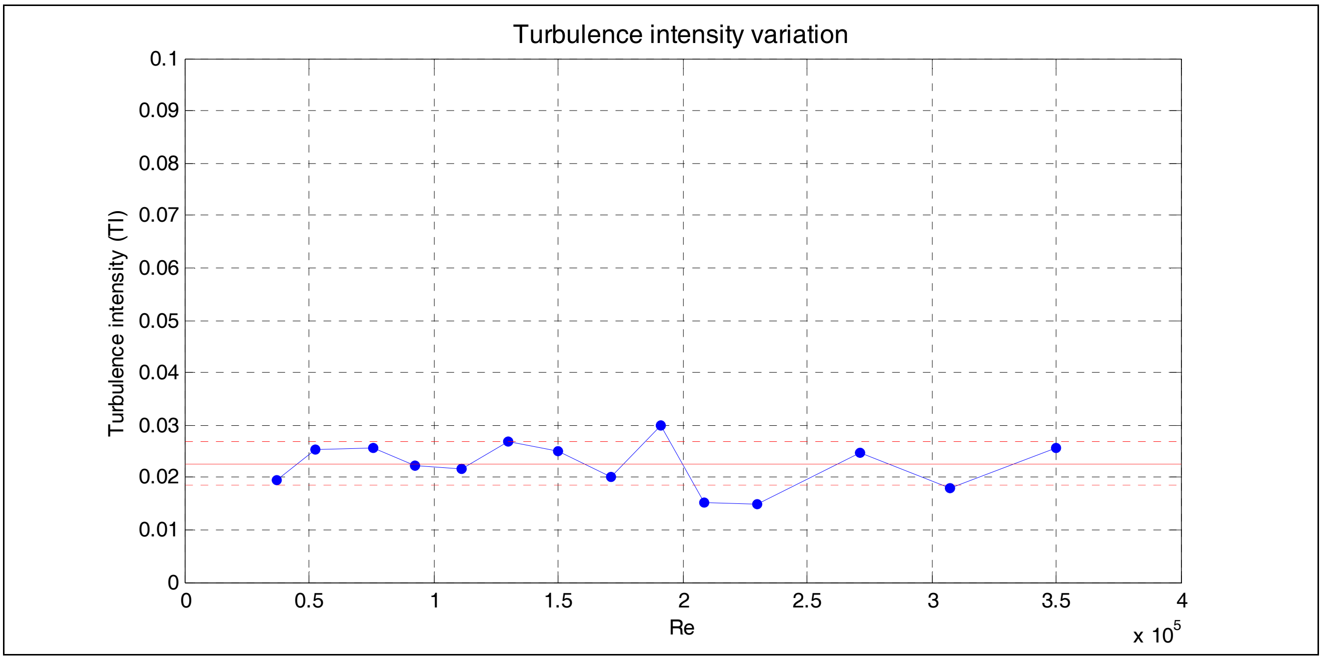

Figure 8: Turbulence intensity variation with Reynolds number (velocities from *0,05m/s* up to *0,5m/s*). The continuous red line represents stands for the mean turbulence intensity $TI = 0{,}022 \pm 0{,}004$, while the dotted red lines represent the standard deviation (order of *20%*).

Cross section profiles

In order to verify the cross section velocity profile, a plane velocity measurement was carried out through the entire width and depth of the test section, resulting in a 49-point measured area. Figure 9 presents the velocity profile obtained for tow conditions of flow rate, corresponding to *0,3m/s* and *0,4m/s*. Velocity contours are presented relatively to the cross section mean velocity, while the length dimensions are nondimensionalized by the test section width.

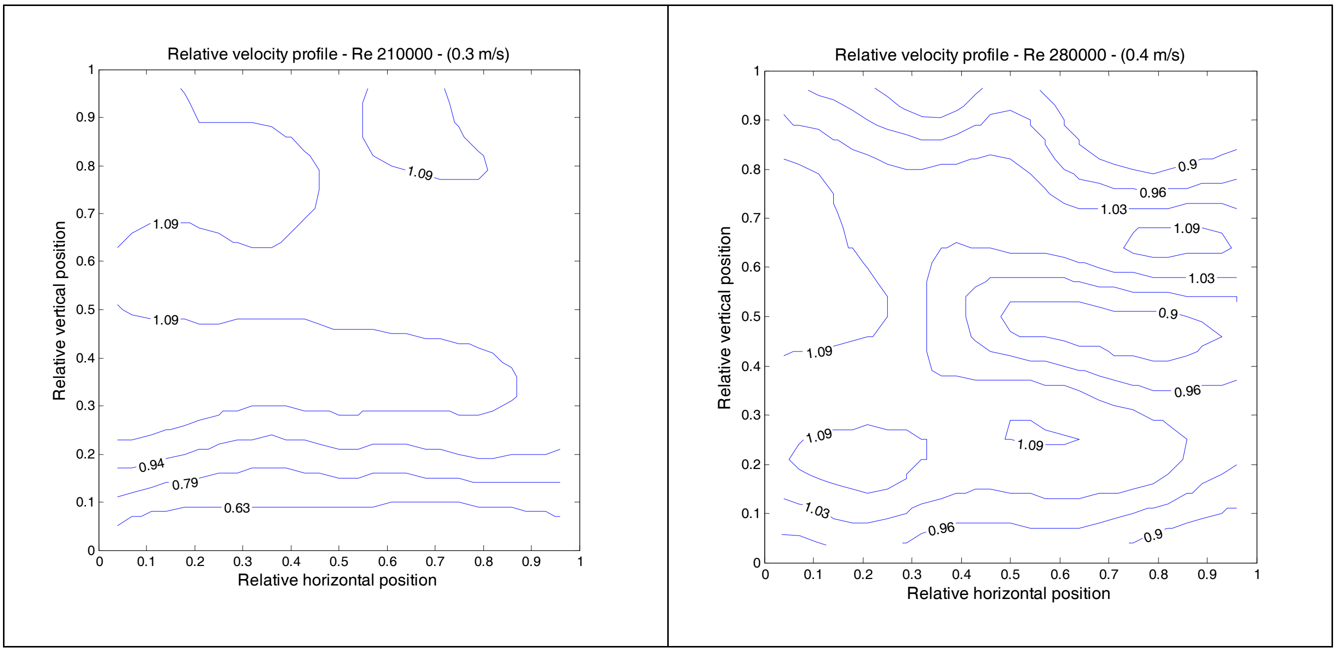

Figure 9: Cross section relative velocity profile for two velocity conditions (*0,3m/s* and *0,4m/s*) presenting velocity contours relative to mean velocity. Length dimensions are nondimensionalized by test section width (*0,7m*).

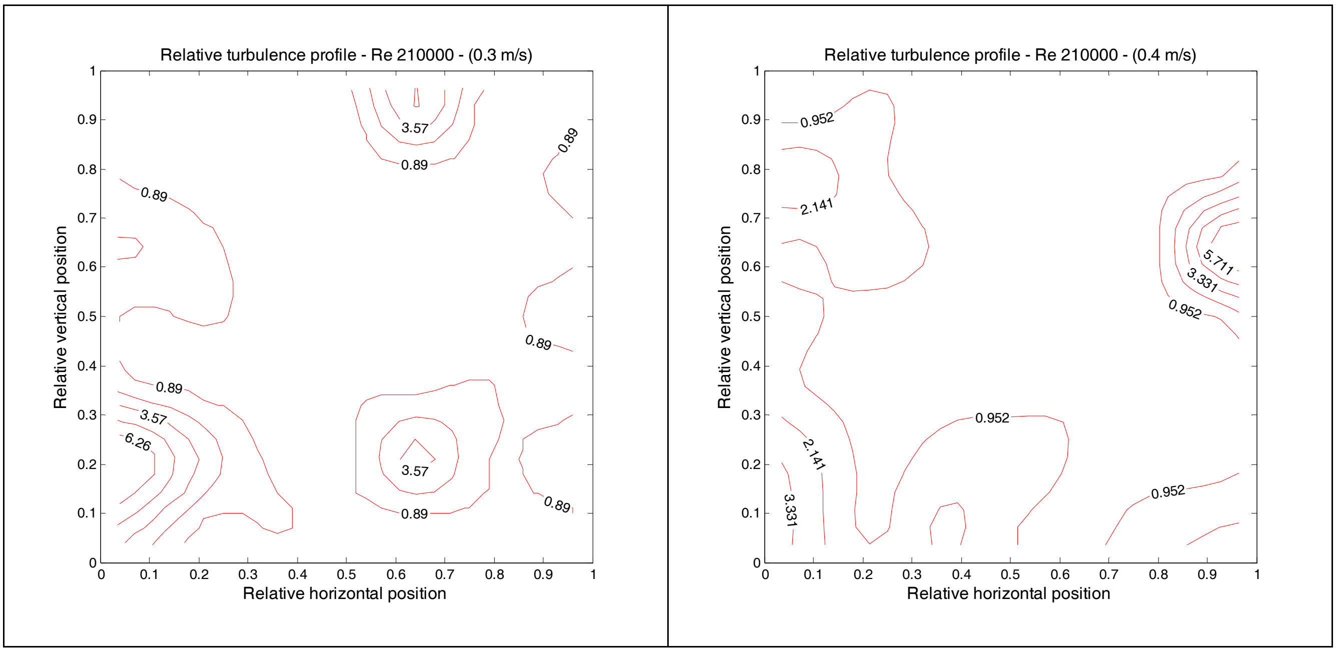

Figure 10: Cross section relative turbulence profile for two velocity conditions (*0,3m/s* and *0,4m/s*) presenting turbulence intensity contours relative to mean turbulence level. Length dimensions are nondimensionalized by test section width (*0,7m*).

For a mean velocity of *0,3m/s* the bottom quarter of the section presents a crescent velocity profile due to the low turbulence boundary layer behaviour. This viscous effect is reduced when the Reynolds number and turbulence level increases for the *0,4m/s* condition. The bottom boundary layer relatively smaller and reaches its mean velocity in just *0,1* depth. In the first case a more plane profile can be seen, while in the second flow condition the velocity profile is closer to the one expected for an open channel. However, the entire section presents a top *10%* asymmetry.

In the same way, the turbulence intensity profile was obtained by the fluctuation of measured velocities in the same 49 points and flow conditions. Figure 10 shows that a higher turbulence level is found near the walls and close to the free surface. In both cases, the order of the mean turbulence intensity is *0,02*.

5 Conclusions

Afterwards, this paper presents the design and construction process of this new facility. Computational Fluid Dynamics (CFD) and scaled models are some of the illustrated techniques employed to accomplish the NDF Circulating Water Channel design. Concerning verification measurements, an overall high-quality result was obtained proving that the test section is ready to be employed for experiments that consider its flow characteristics, such as velocity and turbulence profiles. More detailed test section verification must be carried in order to obtain refined profiles. In addition, velocity profiles measured in different positions through the section length and with different water levels are important to verify the flow quality along the channel. This achievement can be made as soon as possible, when the traverse system became fully operational. Consequently, the flow quality can now be improved by correcting some parameters in the source tube and screen mesh density. New investigations using numerical simulations and scaled models will be employed in future works. Adjusting these properties, velocity profile uniformity and turbulence intensity distribution are expected to smooth. In summary, the NDF Circulating Water Channel presented to be a versatile facility with a satisfactory flow quality test section for hydrodynamics experiments in general.

6 References

Barlow, J.B., Rae, W.H., Pope, A.;1999; *Low-speed wind tunnel testing*; 3rd edition; John Wiley & Sons.

Bell, J.H., Mehta, R.D.; 1989; *Boundary-layer predictions for small low-speed contractions*; AIAA Journal; v. 27, n.3, p.372-374.

Bruun, H.H.; 1995; *Hot-wire anemometry: Principles and signal analysis*; Oxford Science Publications.

Fang, F.; 1997; *A design method for contraction with square and sections*; ASME Journal of Fluids Engineering; v.119, p. 454-458.

Fang, F., Chen, J.C., Hong, Y.T.; 2001; *Experimental and analytical evaluations of flow in a square-to-square wind tunnel contraction*; Journal of Wind Engineering and Industrial Aerodynamics; v. 89, p. 247-262.

Gordon, R., Imbabi, M.; 1998; *CFD simulation and experimental validation of a new closed circuit wind/water tunnel design*; ASME Journal of Fluids Engineering; v. 120, p. 311-318.

Mehta, R.D., Bradshaw, P.; 1979; *Design rules for small low speed wind tunnels*; Aeronautical Journal of the Royal Aeronautical Society; p.443-449.

Morel, T.; 1975; *Comprehensive design of axisymmetric wind tunnel contractions*; ASME Journal of Fluids Engineering; v. 97, p. 225-233.

Morel, T.; 1977; *Design of two-dimensional wind tunnel contractions*; ASME Journal of Fluids Engineering; v. 99, p. 371-377.

SNAJ; 1986; *Directory of ship hydrodynamics research laboratories in Japan*; 2nd edition; Japan Towing Tank Committee; The Society of Naval Architects of Japan.

Wolf, T.; 1995; *Design of a variable contraction for a full-scale automotive wind tunnel*; Journal of Wind Engineering and Insutrial Aerodynamics; v. 56, p. 1-21.

7 Responsibility notice

The authors is are the only responsible for the printed material included in this paper.