DESIGN AND OPTIMIZATION OF A GENERATOR FOR ROTATING FLOWS

By:

Nerea Vidal Franco

1 DE MARZO DE 2019

FORWIND - ZENTRUM FÜR WINDENERGIEFORSCHUNG

Abstract

The study of the interaction of wind flows in different conditions with wind generators has its importance in energy production and reduction of loads on structures. Rotating flows allow for studying the impact of wind from different directions or finding the optimal angle of attack on blades. Throughout research on the topic, it was found that there aren’t studies or devices designed for rotating flow generation. Starting from a wind tunnel, available in the laboratory of ForWind Institute, the aim is to add a device capable of generating the mentioned rotating flows.

In this thesis a rotating flow generator device is developed with the aim to contribute in future wind flow research. The whole process of optimization, in which different blade configurations were tested to get a flow deviation as close as possible to the theoretical maximum value, is explained. The effect on the main flow of different shapes, lengths and numbers of blades was measured with a DANTEC X-Wire in order to get the velocity in both components of X-Y plane. Once the optimization of the device was achieved, a characterization of the flow while increasing the rotational speed is shown to prove the efficiency of the device. Finally, some recommendations and improvements are exposed for later uses of the device.

Contents

List of figures

1. Introduction

Wind energy has established as the most powerful renewable energy source for investments nowadays. About 100 years of research and development made the wind sector an efficient and reliable technology. In the last decade, the wind sector has been experienced an exponential growth all around the world, overtaking stable and ancient technologies such as coal electric generation. According to the Global Wind Energy Council (GWEC) [1], although the global annual installed wind capacity is not exceeding the previous numbers every year (Fig. 1), the global cumulative installed capacity shows an exponential increment from 2001 to 2017 (Fig. 2), expecting the same tendency in later years.

The main wind power market remains in Europe, North America and Asia. In Europe, Germany is heading the most cumulative wind capacity, as well as new installations last years, followed by United Kingdom and France [1, 2]. Spain remains in the second position as the country with the most wind energy capacity of the continent. North America continues growing in the wind market with the influence of US as the second largest market in the world after China, in terms of total installed capacity, despite Canadian wind industry is coming to a standstill. Asia keeps, for the ninth year in a row, the leadership as the biggest wind market due to the high new installations and accumulative capacity of China during last years. India is also exerting a big influence, breaking in 2017 the record year of installations [1] and showing a stable tendency for incoming years.

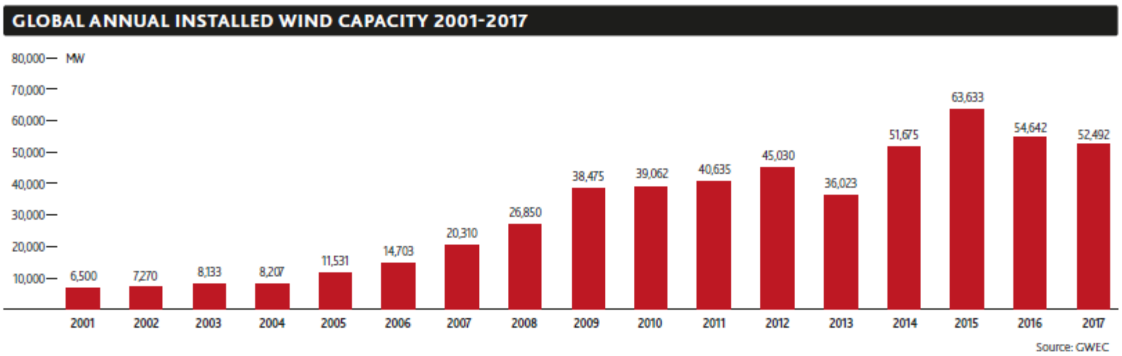


Fig. 1. Global annual installed wind capacity 2001-2017 by [1]

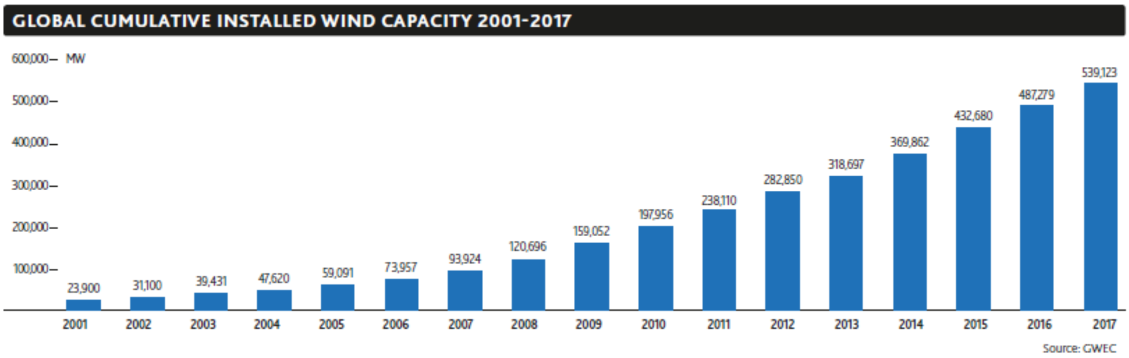


Fig. 2. Global cumulative installed wind capacity 2001-2017 by [1]

The same rising trend is predicted for next years, pushed forward by 2020's targets. Although, in some developed countries, onshore wind energy is almost becoming saturated, offshore energy starts increasing the number of new installations. In addition, countries and regions such as Argentina, Brazil or North Africa which hadn't had until now an important impact in wind installation statistics, start running up their investments in renewable energies. Therefore, a possible deceleration in the number of projects of some of the leader countries will be highly compensated by new potencies.

Regarding the foresight, a continuous improvement of renewable energy technologies is expected. The aim for increasing efficiency and reducing prices will be strengthened by an increment in the number of new projects. In the wind sector, constant researches and developments of new technologies are taking place. The highest efforts are focused on several topics as improving aerodynamics structure of wind turbine components, aiming a better exploitation of wind resource. Finding resilient and lighter materials capable of supporting bigger structures than the actual ones and easier to transport and move. As well as studying the behavior of wind during and after the interaction with wind turbines, effects on the structure, turbulences, etc. Therefore, bigger and powerful devices are expected, allowing for higher power generation wind farms without enlarging the available area.

A topic related to wind behavior and how it affect to structures is its interaction with wind generators. ForWind institute, where this thesis has been developed, has a long experience in small scale turbulence research. The studies of turbulence are made by the stochastic, CFD and experimental groups. The latter, carry out its experiments through three wind tunnel of difference sizes, which allow a lot of versatility when it comes to test wind turbines, new improvements in anemometers or active grids able to impress specific turbulent structures on the flow, in order to expand knowledge in the field of turbulence and its effects on energy production and loads on structures.

This thesis aims to contribute in wind research, providing an initial prototype to add after the outlet of the smallest wind tunnel of the laboratory, which allows for generating rotating flows when changing the outgoing flow. Hence, once a final design of the model is achieved, it would be helpful for speeding up and carrying out future investigations in the field, such as impact of rotating flows on blades and study of angles of attack. For better understanding, in the next section it is explained the theoretical background which lead us to develop this device aiming to generate rotating flows.

1.1. Theoretical background

Initially, our system departs from a small wind tunnel with a square nozzle of 10,5x10,5 cm. The purpose of the thesis is to add a pipe at the nozzle in which the outgoing flow will pass through, modifying, afterwards, the direction when it rotates.

The project is based on the premise that a fluid in contact with a solid adopts the same velocity in its nearest layers. Therefore, when the fluid goes through the pipe in a stationary state, due to the non-slip condition, the adjacent fluid layers have a value of zero velocity, increasing with the distance from the fluid to the pipe wall. In the same way, when the solid has a specific velocity the contiguous layers adopt the same velocity value. In the other hand, it is also considered that the outgoing flow from the wind tunnel has some turbulence. Turbulent flows are known for having a shear stress higher than laminar flows [3], and in consequence, the return to the initial velocities values is slower. Fig 3 illustrates the difference between the shear stress layer of a laminar flow and a turbulent flow, it is appreciated that the shear stress in the turbulent fluid is larger than the other, even though later the return to initial values is faster. Therefore, this property of turbulent conditions will be useful in changing the outflow direction of the wind tunnel, since the fluid will acquire easily the velocity and direction indicated by the pipe when it rotates.

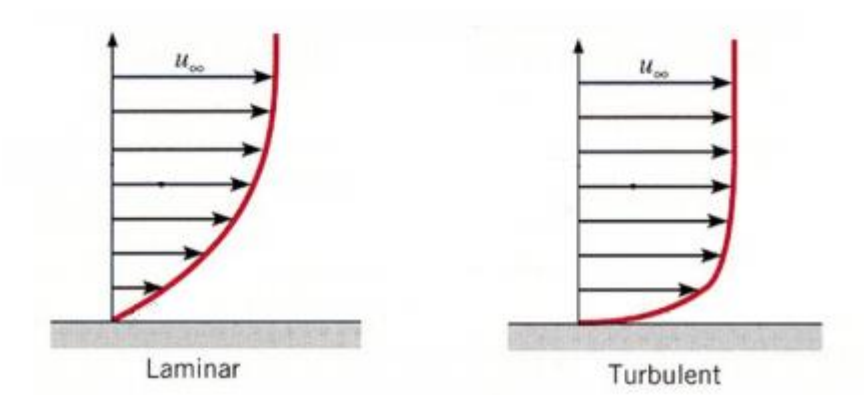


Fig. 3. Comparison between shear stress of laminar and turbulent flow [8]

The project also takes into account the Coriolis Effect [4], which is an inertial force that creates a relative acceleration in a body which is in a rotational system, taken the latter as a reference. For a better understanding, the concept is illustrated in Fig 4. An object or fluid, which is over a rotating surface, is thrown in a straight line from O to Po. Nevertheless, since the surface is rotating, it imposes the velocity to the thrown object, in addition to the velocity of the fluid itself when it is propelled in the straight direction. The fluid adopts a curved path, O to P, from the reference system point of view. Therefore, the Coriolis Effect explains, from the point of view of the rotating system, the acceleration of the body. This effect is also considered in other studies where a fluid passes through a rotating duct or other kind of rotating systems [6, 7]. In case of this thesis work, some blades are added to the setup in order to increase the effect of this two phenomena on changing wind flow direction.

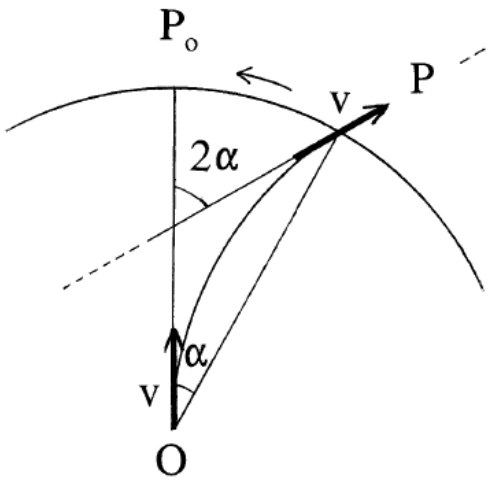


Fig. 4. Coriolis effect [4]

There exist several studies about interactions between fluids and cylinders, such as flow between two rotating concentric cylinders or the effect of a rotating cylinder in which a liquid flows through the outer part, whose applications seem more common in the current industry [5]. However, similar cases related to the generation of rotational flows, such as the one covered by this project, were not found. Therefore, comparison with results obtained in similar studies were not possible.

2. Aim of the project

As mentioned briefly above, the objective of this research is to build a device able to modify the wind direction in order to generate rotating flows and, therefore, getting a different wind flow configuration coming out of the wind tunnel. Thereby, studies about the influence of rotating flows on systems such as blades or turbines can be performed.

The development process consists of assembling and optimizing a rotating device downstream of a wind tunnel. The setup is composed of a set of blades within a pipe where wind passes through while rotating. Along this project, an optimization of the inner blades was carried out, as well as a characterization of the outgoing flow, in order to know the impact of the blades on the wind flow. For the optimization two components of the velocity were measured in the laboratory: the main flux in the x direction, also named as “u”, and the deviated flux in the y axis, known as ”v”. The aim of the experiments was to achieve the highest value in the “v” direction. Therefore, theoretical values were calculated to compare with the data obtained in the laboratory. The equation used to get the maximum values achievable is shown in equation (1), where ω is the angular velocity in rad/s and R the distance from the axis of the pipe. Equation (1) gives the highest value of “v” velocity for a certain point of the pipe and for a specific angular rotation.

$$v = \omega \cdot R \qquad (1)$$

Table 1 shows the theoretical values calculated for the positions measured in the laboratory. Thereby, it is possible to make a comparison within the highest possible velocity value of the deflected flow and the data obtained in the experiments, in order to choose the configuration which approaches the most to the maximum.

Table 1. Theoretical values of flow velocity in the pipe

Distance	5 cm	3 cm
Angular velocity (rpm)	Max velocity (m/s)	
100	0,340	0,146
200	0,681	0,292
300	1,021	0,438
600	2,042	0,875
700	2,382	1,021
800	2,723	1,167

In the process of optimization, different shapes, numbers and lengths of blades were tested to get a setup with the highest deflected flow achievable. The influence of blades on wind behavior was studied for different values of angular velocity of the setup. Hence, an overview of wind performance for different operational modes of the device is provided in this thesis.

Throughout this thesis, the optimization and measurement process is presented, which shows the different tested options for changing the wind flow and provides an explanation of the results. The characterization of the flow while the final configuration works at different angular velocities will be shown as well. Finally, a conclusion of the final design will be shown, as well as advice for its future use and improvement.

3. Experimental set-up

In this section the devices used for running the experiments are described, as well as the optimization process followed along with the results obtained. Each process includes a description of the operational modes used for carrying out the experiments and the measuring positions, which became more accurate as the optimization progressed.

3.1. Setup description

- Wind tunnel

The setup is initially composed by the small wind tunnel to which the developed device is added. The wind tunnel used to carry out the experiments has a 10,5 cm square outlet nozzle. It is controlled by a power supply which gives an input voltage signal in order to get a specific wind velocity. In Fig.5 is shown the relation between some of the input voltage signals and the wind velocities obtained. Throughout the experiments, a constant wind velocity of 4,5 m/s was provided with the aim of comparing the different blades setups.

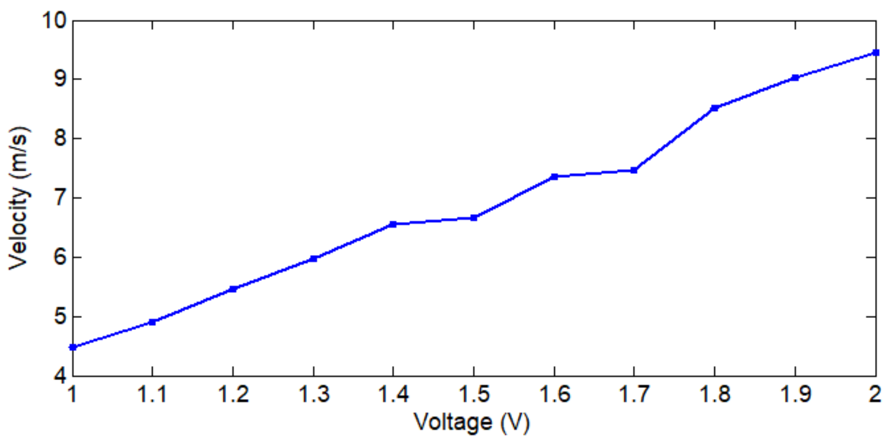
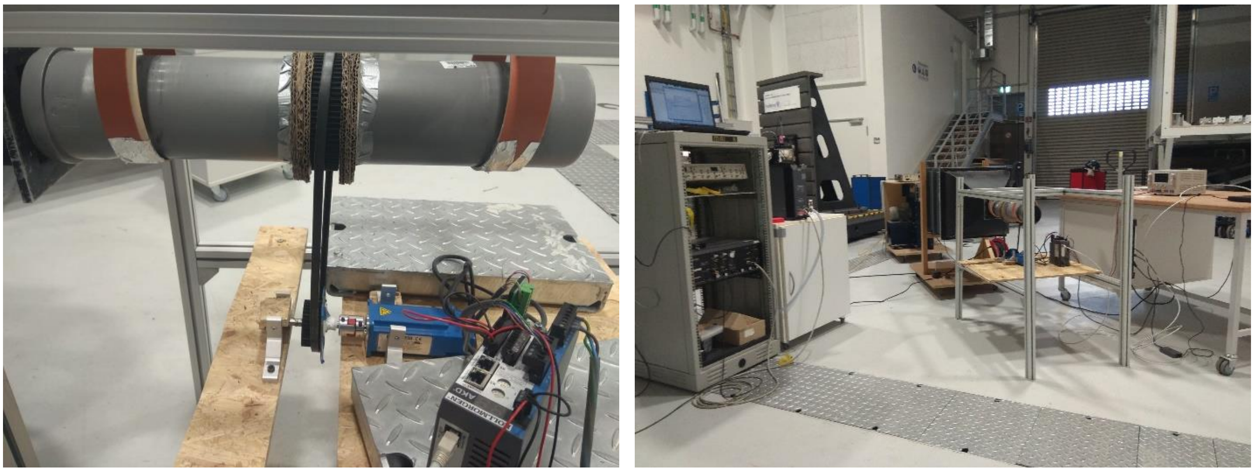


Fig. 5. Relation between applied voltage and output velocity in the wind tunnel

- Rotating flows generator

The built device assembled to the wind tunnel in order to generate rotating flows is mainly composed by a pipe, where the wind flows through, a motor, which makes the pipe rotate, and a frame that holds the device. The frame consists in a metal bar structure of two levels. At the bottom level, a Kollmorgen VLM22C-DLNC-00 motor brand powered with 24V is placed. Connected to the shaft, there are a 3D printed gear of 6 cm of diameter and a bearing on the tip. The purpose of the gear is to facilitate the connection between the engine and the pipe, adding a V-belt as connector. Regarding the bearing, its aim is to hold the motor shaft in a straight line position while allowing it to rotate. The bearing is held by a piece of wood at the top to prevent it from lifting due to the tension created by the connection between the pipe and the motor. At the top level, the pipe is hold by two rings which allow a free rotation. The pipe has a length of 50 cm and a diameter of 10,5 cm, which fits the exit of the wind tunnel very precisely despite of having different shapes. Around the outer part of the tube there is a 11,5 cm diameter gear connected to the motor via the V-belt. Inside the pipe different types of blades were placed for carrying out the study. For further comprehension of the explanation a photo of the device is shown in Fig. 6.



(a) (b)

Fig. 6. (a) Side view of the rotating flow generator. (b) Rotating flow generator with wind tunnel

- Hot-wire anemometer

The velocity of both flow directions was measured with a hot wire anemometer. The probe type used along the experiments is a DANTEC X-wire 55P61. Calibrations were carried out in a velocity range between 2 m/s and 15 m/s, since the measurements were made at 4,5 m/s. Another reason is the reduction of the error of the directional calibration, made between 40º and -40º at 10 m/s. The ambient temperature during all processes varied among 19ºC and 20ºC. The program used to run the calibration was StreamWare Pro v 6.0.

3.2. Optimization and characterization process

In this section, the motivation to carry out each experiment, the different characteristics and operations followed during the measurement processes and the results obtained along all optimization and characterization steps are presented.

- Study of the influence of the pipe on wind flow

The first experiment was performed to investigate the effect of the pipe on the wind flow. Hence, no blades were added to the setup. To check the influence of the built device on the wind flow two currents of wind were measured, the main flow in “x” axis and the deflected flow in “y” axis. In that way, it could be observed if there were changes in the deflected flow or, otherwise, if a constant flow was measured without any impact from the device.

A single position was chosen for the first measurement. The position was assigned where the wind flow deviation was considered to be maximum, close to the boundary of the pipe at a distance of 5 cm from the center, as it is displayed in Fig.7. The operational mode of the device worked at angular velocities of 100 rpm, 200 rpm and 300 rpm.

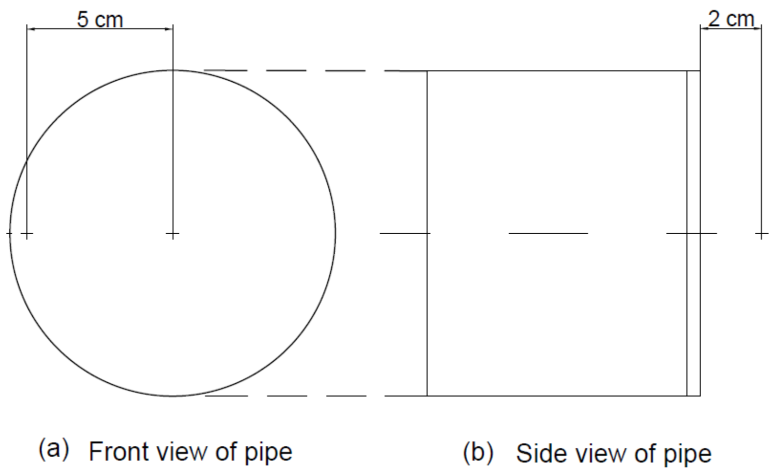


Fig. 7. (a) Distance from center of pipe. (b) Distance from pipe

The results obtained are shown in Fig. 8. It is observed that the influence of the pipe on the flow is non-existent, since the flow remains almost constant. There are some fluctuations in the values due to turbulences inside the pipe that may change the flow direction. Therefore, it was verified that additional blades in the setup were required. Besides, a constant offset in ‘v’ velocity were noticed, which should be close to cero due to uncertainties in the positioning of the X-Wire. The arrangement of the probe was made manually therefore it is not possible to place the anemometer completely horizontal. Also, an offset in the calibration might happen.

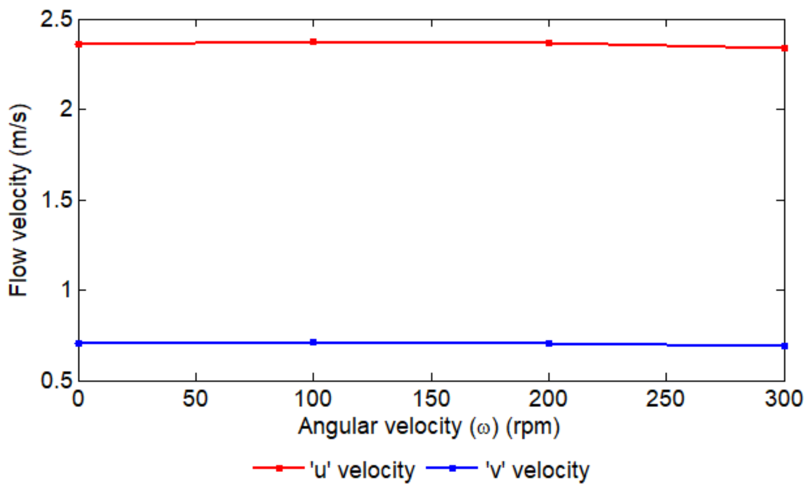


Fig. 8. Velocities in "u" and "v" direction without adding blades

- Blades shape optimization

Once it has been ascertained that the smooth pipe has no influence on the flow, different blade shapes were tested in order to find the most effective configuration in flow deviation. The first configuration consisted of four blades, 10 cm long and 3 cm high, placed in one end of the pipe and arranged at an angle of 90º to each other without reaching the center of the pipe. The other setup tested was a 4-blade cross shape with a length of 10 cm in a 90º disposition.

The measurements were made for 2 positions of the hot wire, whose distance from the center of the pipe is shown in Fig. 9. One of the chosen positions was located close to the boundary of the pipe, where the deviation of the flow was estimated to be maximum and another in between the center and the border of the pipe. These positions were measured at two different distances from the pipe boundary in order to check how the influence on wind flow in the 'y' axis progressed. The operational mode of the device worked at angular velocities of 100 rpm, 200 rpm and 300 rpm.

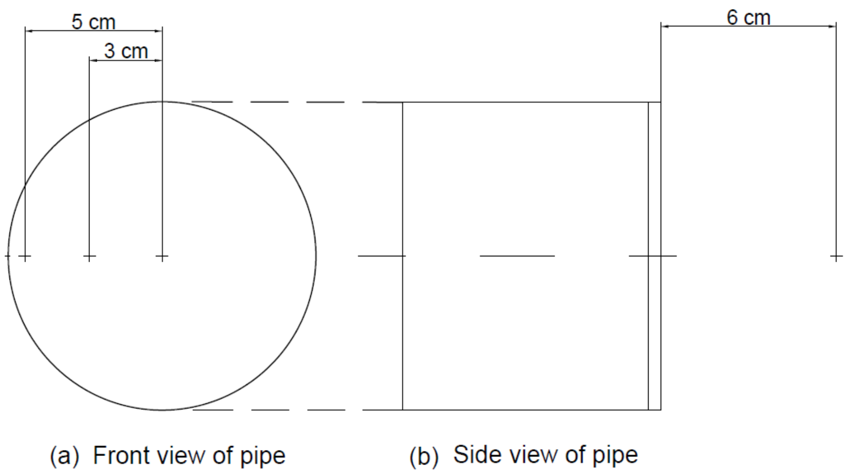


Fig. 9. (a) Distance from center of pipe. (b) Distance from pipe

The comparison between the effect of the single blades and the X shape on the wind flow, together with the maximum achievable theoretical values, is shown in Fig 10. It is observed in the graphs that the wind deviation of the cross shape was more constant than that of the single blades. In Fig. 10 (a), in the position where the influence of both blades was estimated higher, the four cross blades exerted a greater impact on the flow, as well as at a certain distance close to the center of the pipe, where the lack of presence of single blades reduced the impact on the wind flow, Fig. 10 (b).

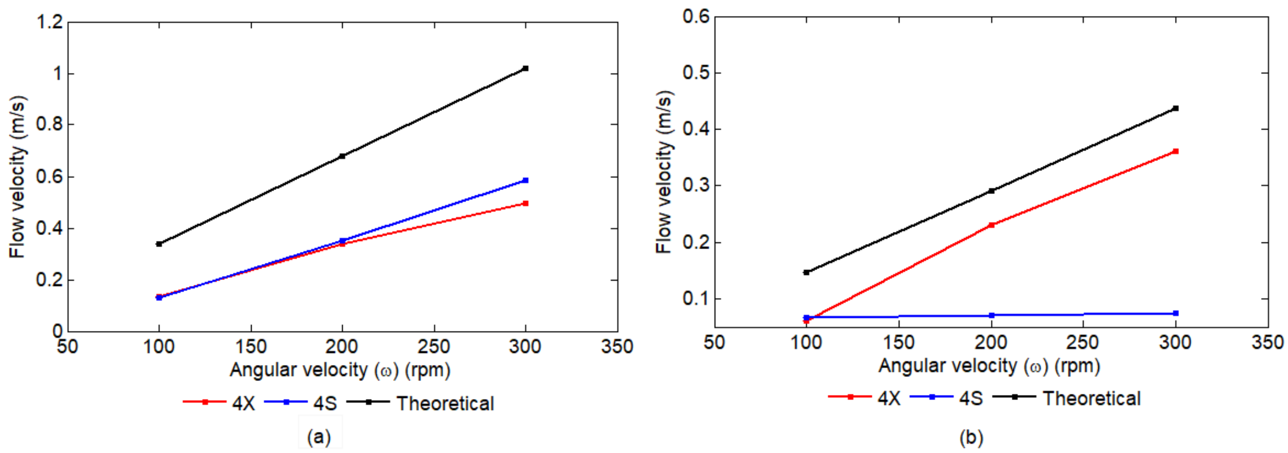


Fig. 10. Comparison between blade shapes at distances of (a) 5 cm from pipe's center - 6 cm from pipe. (b) 3 cm from pipe's center - 6 cm from pipe

After studying the results obtained in the blades shape optimization, it was concluded that the four cross model setup was the optimal of the options tested. Hence, an optimization of the chosen model was made through next experiments.

- Number of blades optimization

After the comparison between the different blade shape configurations and getting the conclusion that an X shape is more effective for the purpose of the project, an optimization of the number of blades in a cross shape was carried out. Therefore, the number of blades was changed until getting a maximum in flow deviation. Three configurations in a cross shape were tested, with different number of blades (2, 3 and 4), all with a length of 10 cm. The angles set between the blades were chosen in such a way that an equal distance was maintained between. Hence, 2 blade cross made an angle of 180º between, 3 blades cross was set in 120º and in case of 4 blade cross the angle between subsequently blades was 90º.

The measurements were carried out at a radial distance close to the boundary layer, where the flow defection is considered to be maximum, and with two different axial distances from the pipe, to observe if the effect on the deflected flow changed. The positions are shown in Fig. 11. The operational mode of the device worked at angular velocities of 100 rpm, 200 rpm, 300 rpm, 600 rpm, 700 rpm and 800 rpm.

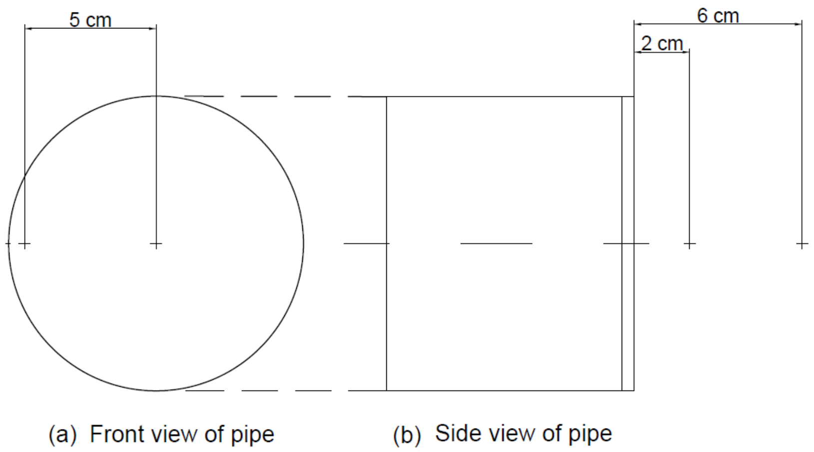


Fig. 11. (a) Distance from center of pipe. (b) Distance from pipe

The results obtained for the different blade shapes are displayed in Fig. 12, where it is noticed that 3 blades was the most effective configuration for flow deviation. Although, in some points shown in Fig. 12 (b) the 2 blade configuration overpassed the efficiency of 3 blades. There is a high difference between both configurations in the position close to the pipe, where the flow deflection of 3 blades was higher and closer to the theoretical values calculated, Fig. 12 (a). Therefore, 3 blades were chosen as the optimum number for the setup.

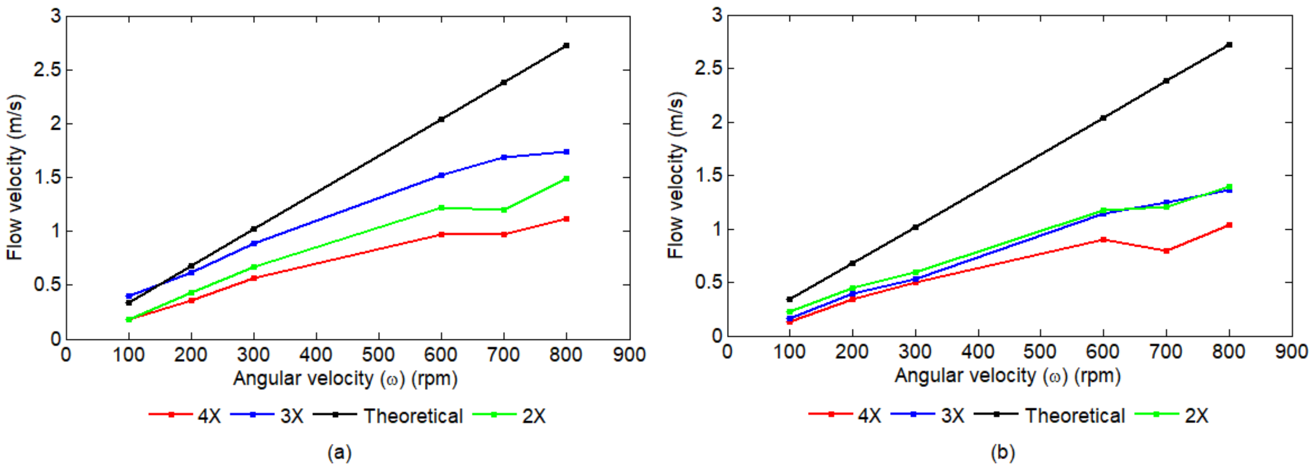


Fig. 12. Comparison between blades number at distances of (a) 5 cm from pipe's center - 2 cm from pipe. (b) 5 cm from pipe's center - 6 cm from pipe

- Length optimization

The last step in the optimization of the setup, after achieving an optimal in number and blades shape, was to study the influence of the blades length on the wind flow. The lengths checked in this optimization process were 5 cm, 10 cm and 20 cm, for a 3 cross blades shape, in order to get a maximum value of flow deviation. Therefore, the different lengths were tested and compared also with the theoretical value so as to check which configuration was closer to the maximum.

The measurement process was carried out under the same conditions as for the optimization of the number of blades. Two positions of the hot wire were chosen at different distance from the pipe, both placed close to the boundary of the pipe, as it is displayed in Fig 11. The operational mode of the device worked at angular velocities of 100 rpm, 200 rpm, 300 rpm, 600 rpm, 700 rpm and 800 rpm.

A comparison between the different lengths and theoretical values is shown in Fig 13. It is observed that a length of 10 cm is the most effective for flow deflection, especially in the position shown in Fig. 13 (a). Although in some points shown in Fig 13 (b) a length of 20 cm seems more effective, the difference in the quantity of flow deviated between both lengths in Fig. 13 (a) is representative. Besides, the magnitude of the flow deflected by 10 cm long blades shown in Fig. 13 (a), compared to the quantity of flow modified by 20 cm long blades displayed in Fig. 13 (b), is higher.

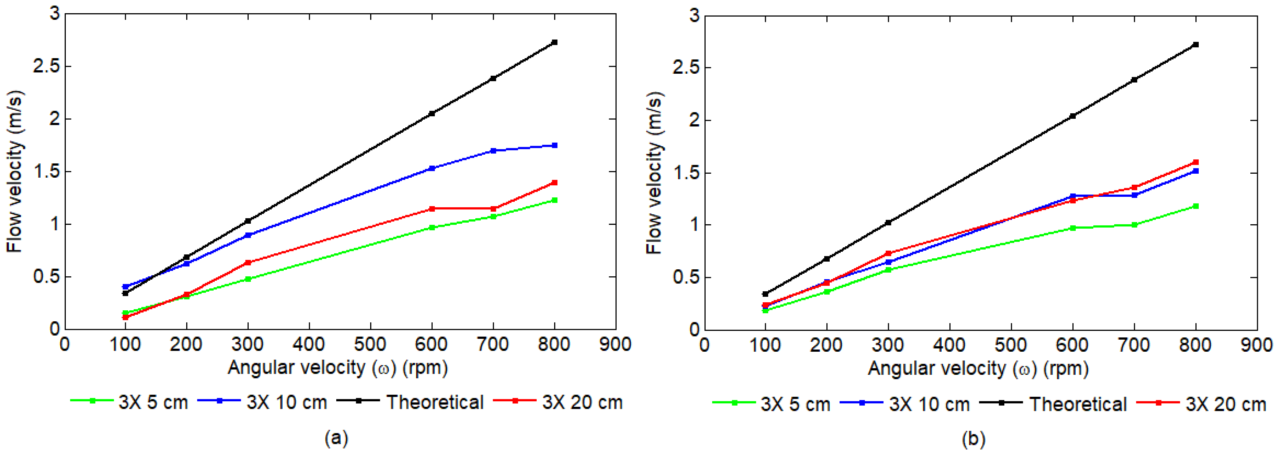


Fig. 13. Comparison between blades length at distances of (a) 5 cm from pipe's center - 2 cm from pipe. (b) 5 cm from pipe's center - 6 cm from pipe

It was observed that the measurements made in this experiment, for the 3 blades 10 cm length cross configuration, were similar in shape and magnitude to the values acquired in the blade number optimization, for the same blade setup, which gives an idea of a good operation of the configuration as well as the measurements taken.

- Characterization processes

The characterization of the flow involved two processes. The first characterization was to investigate the flow behavior on both sides of the pipe in order to test asymmetric wind directions during deflection, as the measurements were made on the left side of the pipe during the entire optimization process. Hence, characterization of the wind flow, along the complete section of the pipe, was made with the aim of proving similar values in symmetric positions but opposite directions and, consequently, verify the processes.

In order to get a better characterization of the wind behavior, more precise measurement positions were taken than in the optimization process. The measurements were taken in one centimeter steps from the center of the pipe to the outer edges, at a constant axial distance of 2 cm away from the pipe, as it is shown in Fig 14. The operational mode of the device worked at angular velocities of 100 rpm, 200 rpm, 300 rpm, 600 rpm, 700 rpm and 800 rpm.

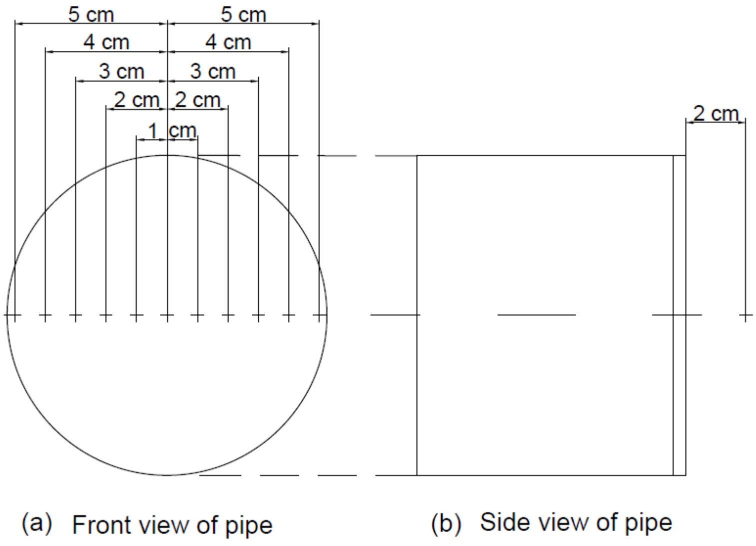


Fig. 14. (a) Distance from center of pipe. (b) Distance from pipe

The results of the first characterization are displayed in Fig. 15, which shows the comparison of both sides in absolute values, where is a close relation between symmetric positions observed. There is a small misalignment between the values, which may be related with the precision of the hot wire anemometer and turbulences in the wind flow. While increasing the rotational speed, an increment in flow deviation can also be noticed.

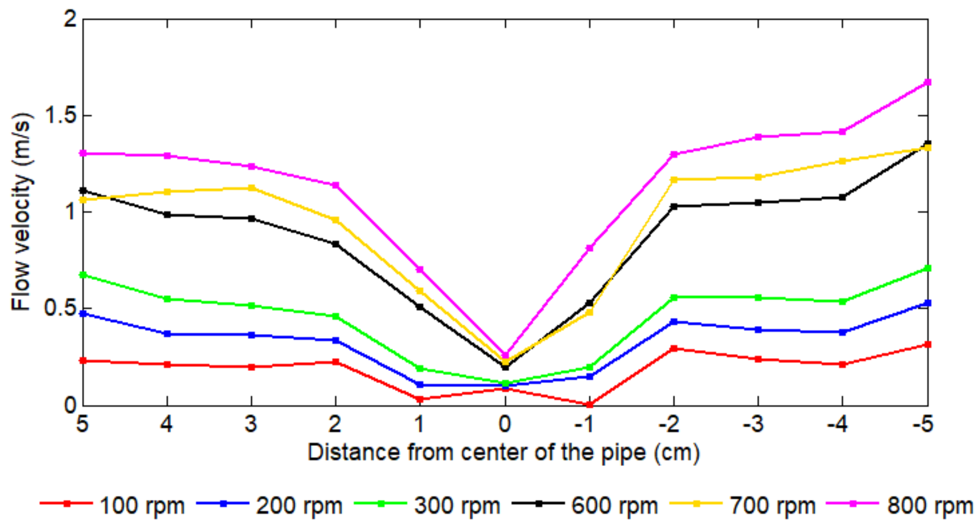


Fig. 15. Comparison between deflected flow at both sides of the pipe (in absolute value)

The second characterization was a study of the wind flow from the center of the pipe until almost 4 cm further from the edge, measured at different distances from the device, in order to characterize its behavior. Both directions of wind were checked, the main flow in 'x' axis and the deflected flow in 'y' axis. Therefore, the aim of the characterization is to see the influence of the device over the wind flow and observe the evolution of the rotating flow.

The measurements were taken in one centimeter steps from the center of the pipe extending 3,75 cm beyond the edge of the pipe, at different axial distances from the pipe of 2cm, 6 cm, 10 cm and 15 cm. A layout of the positions is shown in Fig. 16. The operational mode of the device worked at angular velocities of 100 rpm, 200 rpm, 300 rpm, 600 rpm, 700 rpm and 800 rpm. In order to show the influence of the rotation of the device, the wind flow behavior was also measured when the device didn't rotate.

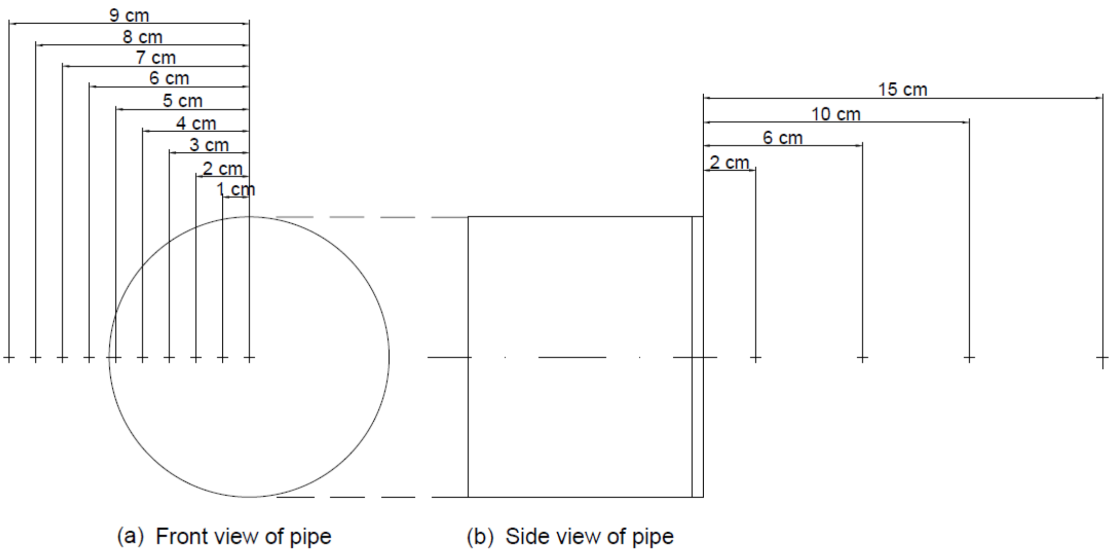
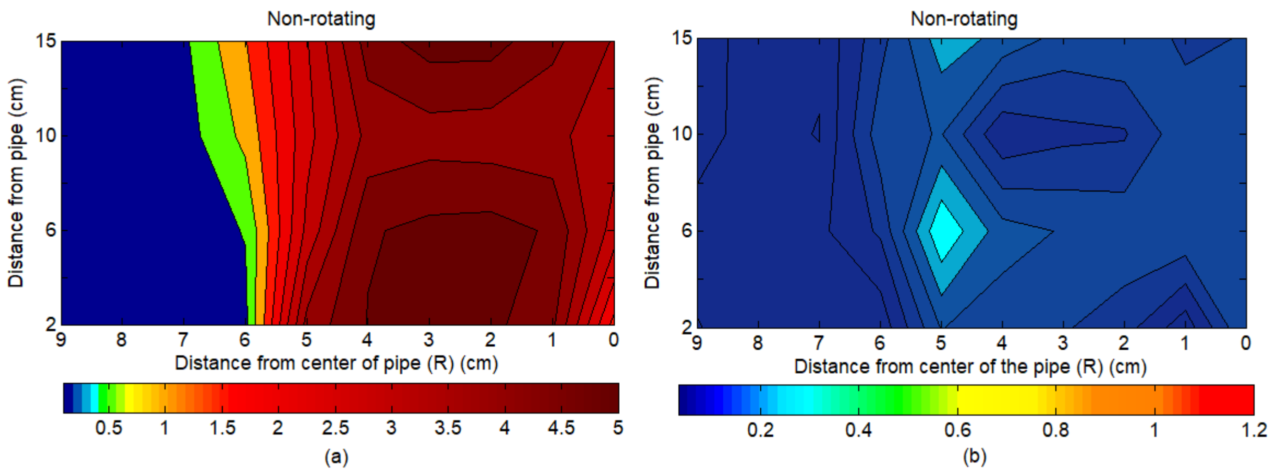
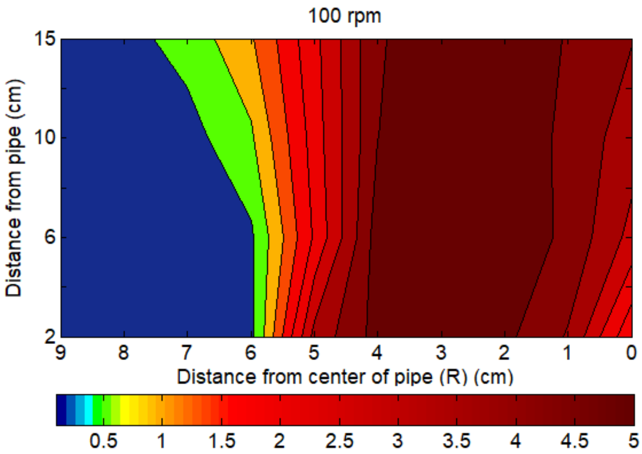
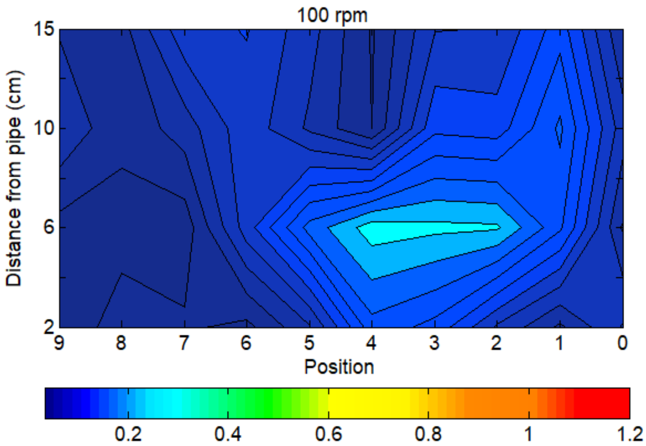


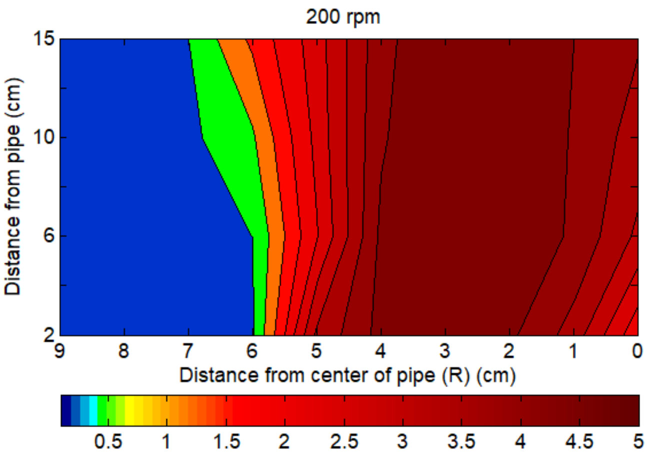
Fig. 16. (a) Distance from center of pipe. (b) Distance from pipe

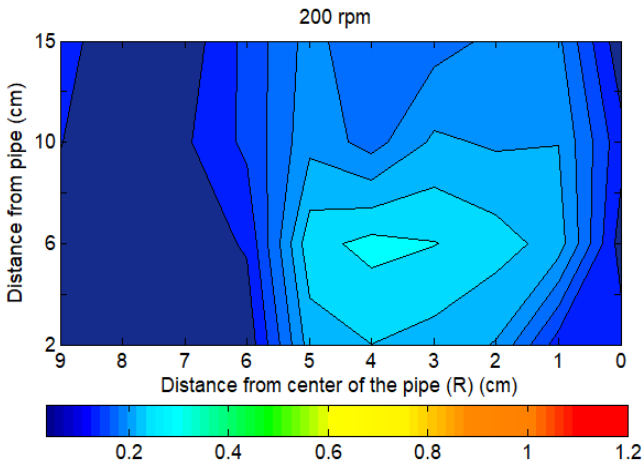
Fig. 17 below shows the flow measured in two columns, in the left side the evolution of the main flow 'x' is represented, while in the right side the quantity of the deflected flux in 'y' axis can be seen.

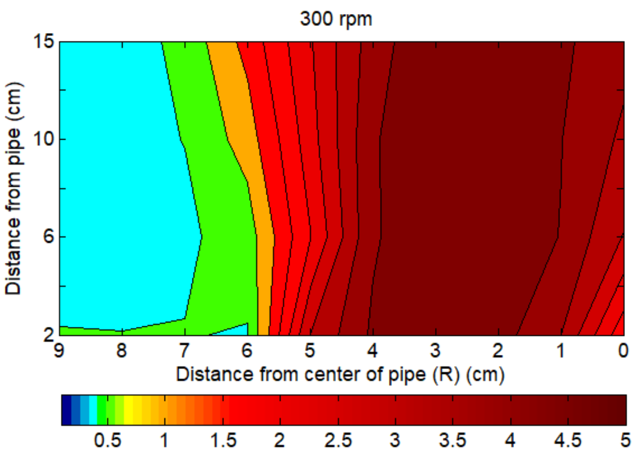


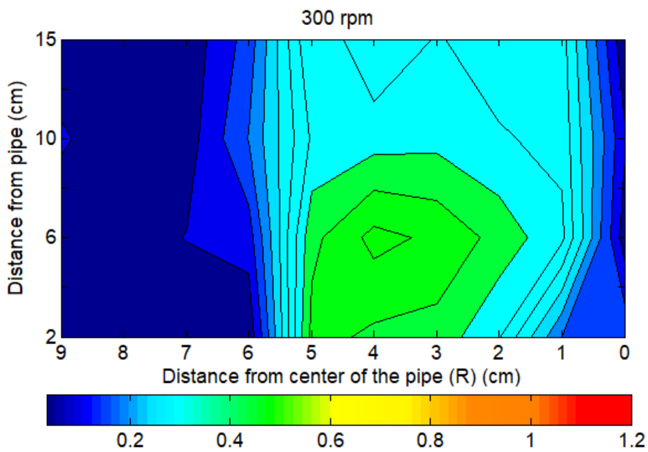

100 rpm
Distance from pipe (cm)
Distance from center of pipe (R) (cm)
(c)


100 rpm
Distance from pipe (cm)
Position
(d)


200 rpm
Distance from pipe (cm)
Distance from center of pipe (R) (cm)
(e)


200 rpm
Distance from pipe (cm)
Distance from center of the pipe (R) (cm)
(f)


300 rpm
Distance from pipe (cm)
Distance from center of pipe (R) (cm)
(g)


300 rpm
Distance from pipe (cm)
Distance from center of the pipe (R) (cm)
(h)

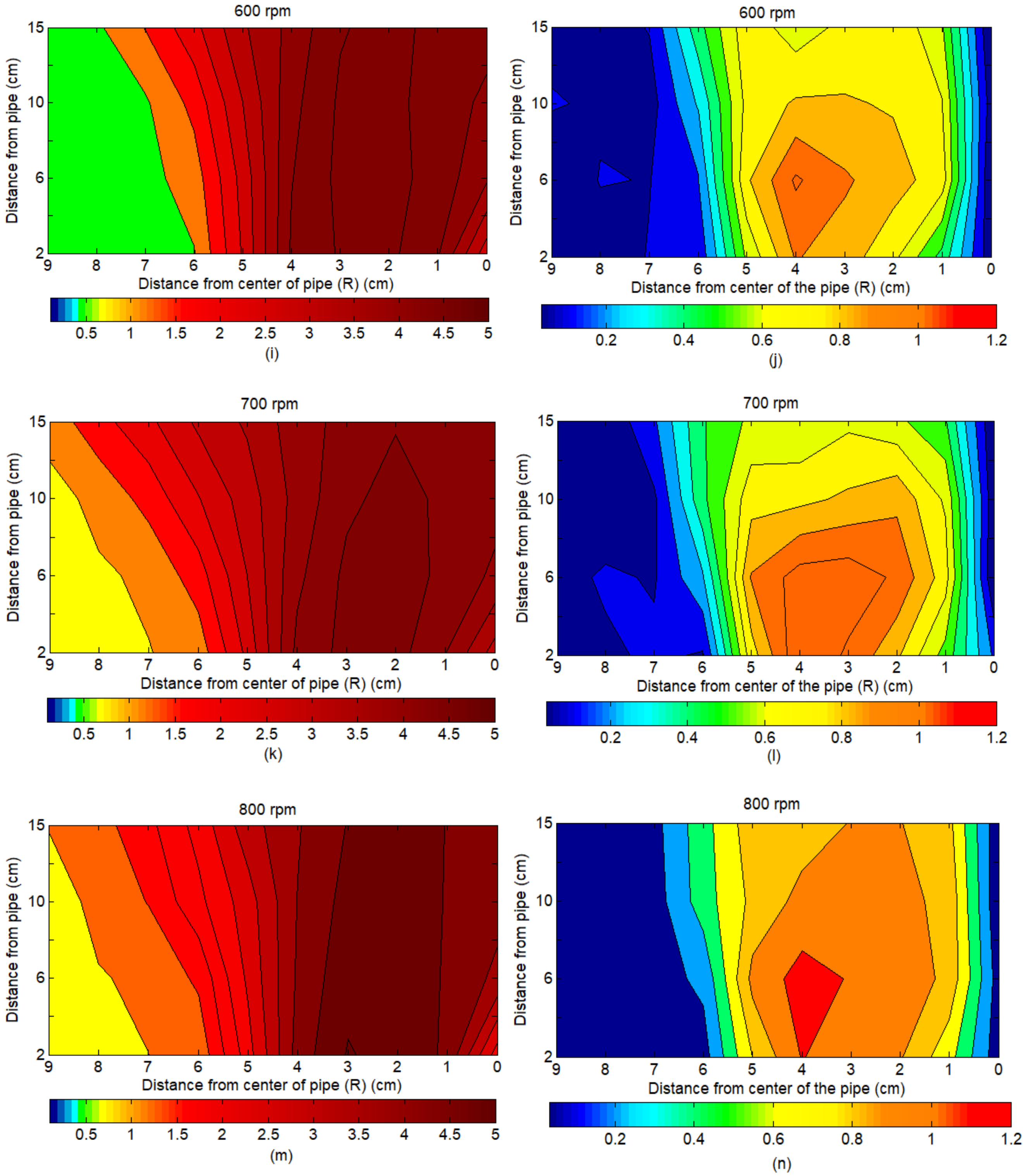


Fig. 17. Characterization of wind flow in "x" axis (left) and "y" axis (right) for rotational speeds of (a) - (b) non-rotating. (c) - (d) 100 rpm. (e) - (f) 200 rpm. (g) - (h) 300 rpm. (i) - (j) 600 rpm. (k) - (l) 700 rpm. (m) - (n) 800 rpm

In the transition of the graphs it can be observed the expansion of the wind due to the influence of the blades. The first figures, (a) and (b), show the behavior of the wind when it passes through the non-rotating pipe. The main flow follows a straight direction, showing a significant change in wind speed between the inside and outside of the pipe. The highest velocities inside the pipe were observed close to the boundary, where the influence of the non-rotating blades decreased, while in the center the junction of the blades reduced the wind speed. The effect of the blades junction on the wind flow can be noticed since it generated a velocity deficit in the center downstream. Regarding the velocity outside the pipe, reaching a value close to zero after 1 cm away from the boundary. In case of the flow in the ‘y’ axis, the values measured are also in the range of zero velocity. Small changes in speed are attributed to the turbulence generated inside the pipe.

After the study of the unaltered flow, its evolution when increasing the rotational speed was investigated. For angular velocities of 100 rpm and 200 rpm, Fig. 17 (c), (d), (e) and (f), it is observed that the main flow behavior barely changed in relation to non-rotational state but experienced an increase in rotational flow expansion in the furthest distances as 10 cm and 15 cm. In the case of the deflected flow, the velocity increased in the closest distances from the pipe. The highest changes in the flow were observed in the area nearer to the pipe boundary, these effects reduced while approaching the core due to the blade connection point. The increasing trend is maintained for 300 rpm, Fig. 17 (h), and the expansion on the main flow is more noticeable, Fig. 17 (g), where the velocity of wind in the surroundings of the pipe started changing its velocity.

When a step further was giving for angular velocity ranges, i.e. between 600 rpm and 800 rpm, changes in wind flow increased considerably. For the velocity "u", x-axis, the generated rotating flow suffered an expansion that was observed beyond the pipe boundaries, measuring speeds of 0,4 m/s at 600 rpm until 1 m/s at 800 rpm, for a distance of 1 cm beyond the radial boundary and 2 cm in axial direction away from the pipe. The same velocity values were achieved at 15 cm in axial direction away from the pipe for a distance 4 cm beyond the radial boundary. Rotational flow expansion in the main flow direction can be seen in Fig. 17 (i), (k) and (m). In case of ‘v’ velocity, ‘y’ axis, wind deflection is even observed at 15 cm axial distance. The largest wind deviation was seen at a radial distance of 5 cm inside the boundary and along 2 cm and 6 cm in axial distance away from the pipe, with increasing angular velocity, an increase in the wind deviation was observed. Changes in flow deviation for rotational speeds of 600 rpm, 700 rpm and 800 rpm can be observed in Fig. 17 (j), (l), (n).

The results from the characterization of the wind flow prove the capability of the built setup to generate rotational flows which can be used for further research.

4. Conclusions

Generation of rotating flows was the aim and achievement of this research. Firstly, a rotating device was designed and built up to pass the wind through. Several blade shapes, in different numbers and lengths were attached and tested, until the optimal setup for both flow deviation and rotating flow effect was reached. Afterwards, the flow generated at different rotational speeds were measured and characterized, when the device rotated and angular velocity rose, an increase of velocity in the y direction and a deflection of the flow were observed.

The wind coming out of the wind tunnel follows a straight direction when the device does not rotate. When the device rotates it generates a rotating flow that expands sideways, beyond the edge of the pipe, an effect that increases with the angular velocity. It was perceived that the generated rotating flow expanded very fast out of the pipe boundary. As a result, it is recommended to limit the sideways expansion while using this device to generate rotating flow for future experiments. Additional structures to block the wind expansion, or the connection of a second pipe are proposed as two possible solutions to minimize the effects of the sideways expansion.

5. References

[1] GLOBAL WIND REPORT. Annual market update 2017. Global Wind Energy Council

[2] Wind in power 2017, Annual combined onshore and offshore wind energy statistics. Wind Europe.

[3] Mechanics of Fluids. Merle C. Potter and David D. Wiggert. 3rd Edition.

[4] How Do We Understand the Coriolis Force? Anders Persson. European Centre for Medium-Range Weather Forecasts, Reading, Berkshire, United Kingdom.

[5] Rotating Flow. Peter R.N. Childs. 1st Edition.

[6] Experimental investigation of a rapidly rotating turbulent duct flow. G.E. Ma°rtensson, J. Gunnarsson, A.V. Johansson, H. Moberg. Sweden, 2002.

[7] Effect of system rotating on turbulent boundary layer flow. Li Haiwang, Jiang Zhiyao, Tao Zhi, You Ruquan, Wu Huijie. China, 2019.

[8] Supersonic Wings with Skin Friction: A Primer on Boundary Layers and Skin Friction. Utah State University.

[9] Introducción a la mecánica de fluidos. MOD 6 Flujo Rotacional y Vorticidad. (Introduction to fluid mechanics. MOD 6 Rotational flow and vorticity). Ing. Jorge Rosasco.