Modeling of irregular shaped objects via pressure gauge in a turbulent flow

Author: Oriol Lujan Saez
Facultat de Física, Universitat de Barcelona, Diagonal 645, 08028 Barcelona, Spain.

Advisor: Dr. Anna Vilà i Arbonès

Abstract: In this project simulations and experimental procedure are used to study the variations in the pressure inside and outside the turbulence of regular objects to make a comparative with irregular objects. The results show that this will be useful in the approximations done in heat transfer problems.

I. INTRODUCTION

In order to study the heat transfer in a system [1], three ways of transferring energy need to the considered:

1. *Conduction.* The heat is transferred when the two (or more) objects are in contact. To calculate the amount of heat transferred the Fourier's Law can be used.

2. *Convection.* The heat is transferred because of the different temperatures between a body and the fluid that is surrounding its surface. There's a law to calculate the amount of heat transferred, the Newton's law of cooling:

$$\dot{Q} = hA\Delta T \tag{1}$$

   where h is the convection constant, A the surface of the body in contact with the fluid and ΔT is the difference between the two temperatures.

3. *Radiation.* The heat is transferred due to an electromagnetic radiation produced by the movement of the charged particles of the body. It can be calculated using the Stefan-Boltzmann law.

In Eq. (1) the main problem is to calculate the convection constant h. This constant is the factor of the equation that correlates the relative movement of the body and the fluid, the properties of the surface of the body, and the properties of the flux with the amount of the heat that has been transferred in this process.

Due to the amount of properties that can affect to this number h, it's very difficult to calculate it in an analytical way. Usually, it is rather obtained after an experimental correlation in order to be able to calculate the number. Nowadays we can use the finite elements method to solve the Navier-Stokes equations and calculate this number, but in a lot of cases getting a solution is quite expensive and takes a lot of time.

This experimental correlation needs to find out a relationship between the Reynolds Number (Re), the Prandtl Number (Pr) and the Nusselt Number (Nu), that can be found in an experiment and is similar to the next equation:

$$Nu(Re, Pr) = ARe^m Pr^n \tag{2}$$

where A, m, n are the coefficients to be determined in an experiment in order to calculate h. We use these magnitudes due to it's relation with h:

$$Nu = \frac{hL}{k_f} \tag{3}$$

where L is the characteristic length and k_f is the thermal conductivity of the fluid.

Nowadays there are many tables with these relationship for different types of flows [2]. It's necessary to differentiate between the laminar flow and the turbulent one because the coefficients A, m, n would not be the same for the different types of flow.

To find out which relationship $Nu(Re, Pr)$ is the best option to use in a problem it is necessary to approximate the object to a simplest shape like a sphere or a cylinder, but in many cases it's difficult to know which is the best option to approximate the irregular shaped object to the simplest one. In the present study, a new experimental method to calculate h for complex objects will be developed, using the hypothesis that inside the turbulence created by the relative movement of an object through a fluid, the pressure is slightly different inside than outside of the turbulence. This hypothesis is presented in Fig. 1:

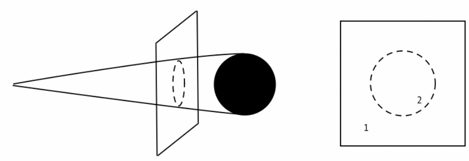

Fig. 1: Figure of the different zones of pressure in a turbulence

As it can be seen in the Fig. 1, there are two zones of interest in the schematic plane. The first one (1) is the one out of the turbulence, the pressure will achieve the value due to the dynamical pressure added to the gravity contribution. The inner one (2) the pressure value will be fluctuating due to the unpredictability of the form of the turbulence with precision. That means that if the system is able to differentiate between pressures in different coplanar points, it will be possible to determinate the shape of the turbulence and compare between other turbulence generated by regular objects.

Following this theory, if a system is capable to calculate the pressure of different points and make a comparison with other pressures (of different objects at the same positions), it would be easier to make a model of the irregular object to a regular one with similar values; later on the system can calculate the parameters and get to a solution of h and calculate the error of the approximation.

II. EXPERIMENTAL DESIGN

A. Sensors

To calculate the difference of pressures between the different zones a pressure gauge can be used, sensors can be placed in a specific place to capture the pressure of a particular point. In order to study different zones, more than one gauge can be applied (five in this particular case). These gauges can be distributed in many geometries that can provide information of points of interest. For this experiment, a geometry that allows the system to differentiate between different shapes of the turbulence is needed, so the best geometry with five sensors is the one showed in Fig. 2:

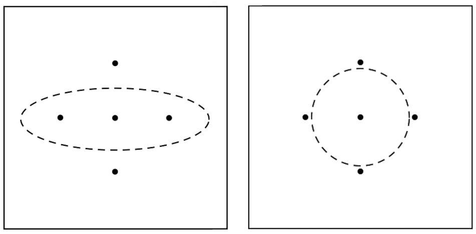

Fig. 2: Geometry of the pressure gauges

This geometry makes easier to differentiate between a circular turbulence (caused by an spherical body) and an elliptical one (caused by a cylindrical body).

These sensors need to be small to detect just the pressure in a specific point, taking into account this necessity, a kind of sensor that uses the pressure variation to change the length of a thin resistor, increasing its resistance, has been chosen [3]. What the system detects is this change of resistance so the most important thing is to determinate the behaviour of the gauge to make a correlation between the pressure applied and the resistance that is being displayed.

To read the data of the pressure gauge, a multimeter reading the resistance while changes can be used, but a better option is to use an Arduino Uno [4]. This system is perfect due to its versatility and because it has five ports to read the data (the same number of sensors as the geometry requires) and it has a pin board that will be used to create the following system in the Figure 3:

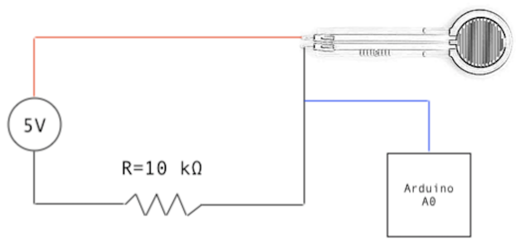

Fig. 3: Schematic of the electric circuit used to acquire the data

As far as the experiment will use five pressure gauge, the circuit will change the number of Arduino Pin's used to acquire the data.

B. Experimental setup

At this point, a recipient able to create a flux is required, with the idea of putting an object inside the fluid to generate a turbulence. To make that possible, a fluid pump is needed to create this movement of the fluid (in this case water), also a diffuser to stabilize the flux that needs to be laminar, in order to have just the turbulence generated by the object avoiding an extra turbulence that would be affecting the data in the measurement of the pressure; finally the object and the sensors that will provide the pressure data. This setup is presented in Fig. 4:

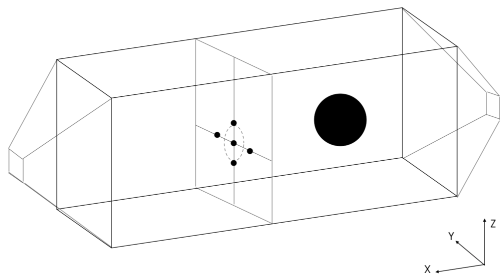

Fig. 4: Schematic of the experimental system where the black body is the object under study, the pump will be connected via tubes to the edges of the recipient and the flux will be flowing from the part near to the object to the opposite one.

C. Methodology

The goal of this project is to create a database of the pressure of different points for different regular objects in order to being able to calculate an approximation of h for non regular objects. For this project, the pressure of the turbulence of two regular objects with very different shapes is the main study, this shapes are a sphere and a cylinder. This regular objects will produce a very different turbulence so the (2) zone in the Fig. 1 will have different shapes for each object.

The first step will be creating an array that will provide the pressure on a certain point so there will be five numbers according to each sensor used to get the pressure data. To create an array of five static numbers it will be necessary to make an approximation and taking the average number of each position, so the array will look like this:

$$P_{s,c} = \left[\sum_{i=1}^{N_1} \frac{p_{1i}}{N_1}, \sum_{i=1}^{N_2} \frac{p_{2i}}{N_2}, \sum_{i=1}^{N_3} \frac{p_{3i}}{N_3}, \sum_{i=1}^{N_4} \frac{p_{4i}}{N_4}, \sum_{i=1}^{N_5} \frac{p_{5i}}{N_5} \right] \quad (4)$$

where N_i is the number data acquired and p_i is the pressure for each position. This equation is useful for both, the sphere and the cylinder. Once the array is calculated for each bodies, the study of an irregular object by calculating its array and make a comparison with the database would be interesting. This analysis will calculate the error between each number (1 ,..., 5) of the two arrays (the irregular object array and one of the database), then calculating the same error with the other array in the database would be the next step. Finally the five numbers of error that minimizes the global error will lead the system to a solution, and the body corresponding to this solution become the optimum approximation of the irregular body. Knowing this error of the approximation the number h with an error associated to this solution can be calculated.

III. SIMULATION

Using CFD (Computational Fluids Dynamics) software it can be modeled any system with an interaction through a fluid. In order to make the results of this experimental procedure more visual and have more precision with the numbers that are in the experiment it will be used a cloud computing software to model this experimental procedure. It will be used the software *SimScale* [6], that allows uploading the 3D figure that will be studied inside the flow of a fluid and make different computations with different parameters like relative velocity, pressure, temperature, ... To make the simulations, the software discriminate the continuous media so it can be solved by finite element analysis [5]. In this case they're going to be studied the following two regular figures and one irregular: sphere, cylinder and a wing profile. A 3D image of this objects is shown in the Figure 5.

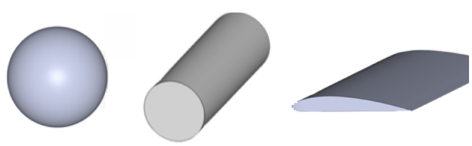

Fig. 5: 3D figures that will be analyzed

Software	Simulation type	Flow type
SimScale	Convective Heat transf.	Laminar
Velocity inlet V_x	Pressure outlet P_o	Gravity g_z
1.5 m/s	0 Pa	-9.8 m/s^2

Table 1: Parameters of the simulation

A. Sphere

In this case, the simulation will describe the behavior of the fluid flowing around a sphere. To set up the simulation, the first step is to establish the parameters that will affect to the results as the velocity of the flow and the pressure at the output. The simulation is going to be for a sphere with a radius of $R = 5$cm. The results for the pressure values for a sphere with the parameters in the Table 1 are shown in the Figure 6:

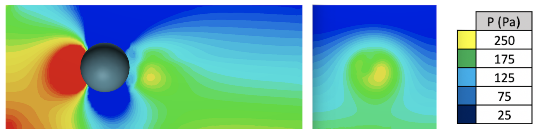

Fig. 6: The left image is the pressure in a XZ plane, the right one is the pressure of a YZ plan after the sphere

As it can be seen at the right image, there is a circular zone where the pressure is slightly different. This demonstrates that the hypothesis of the Fig. 1 is a good approximation of what is happening with the pressure.

B. Cylinder

The cylinder is the other regular shape different enough of the sphere that will be simulated. To model this figure a 3D cylinder with $R = 5$cm and a height of $H = 15$cm with the larger direction perpendicular to the flow direction will be used.

Now, following the hypothesis of the Fig. 1, the shape of the turbulence behind the object will have a larger shape, similar to an ellipse. Using the same parameters as the Table 1, the pressure behaviour for the cylinder is showed in Fig. 7:

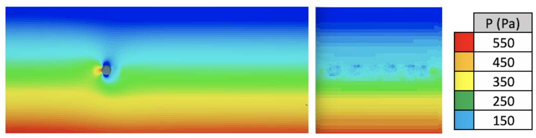

Fig. 7: The left image is the pressure in a XZ plane, the right one is the pressure of a YZ plane after the cylinder

Where again the hypothesis is correct, on the right image it can be seen a large pattern with a different color that indicates that the pressure inside is quite different. Here, the effect of the gravity is quite larger than the simulation before, it can be appreciated by the horizontal lines of different colours.

C. Wing profile

In order to study an irregular shape different than the regulars that have been studied, one of the objects with more interest of being simulated is the wing profile. The dimensions are: $20cm$ wide, $5cm$ height and $30cm$ long. The results for this simulation are presented in the Fig. 8:

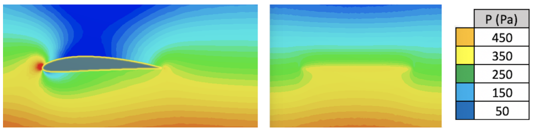

Fig. 8: In this image it can be seen the pressure distribution in a wing profile (left)(XZ plane), the shape of the turbulence (right)(YZ plane)

It is interesting to remark that the pressure below the wing is higher than over it and that is what causes the elevation force allowing the planes to fly. Is shown that the shape of the turbulence is quite similar as the cylinder one, but thinner and larger. In this case the approximation would be the cylinder attending to the shape of the turbulence and the pressure inside and outside this zones. As the wing is going through the fluid in one direction, the effective area is just the frontal one as the cylinder, that's why it can be approximated to a cylinder with the similar effective area.

IV. EXPERIMENTAL PROCEDURE

A. Gauge characterization

In order to establish the relationship between the pressure applied to the sensor and the data acquired, it is important to perform different experiments to calibrate the device.

1. *Experimental design 1*

In this simplest design, weights (coins of 20 cents) will be put successively onto the sensors to add equal amounts of pressure to the gauge and determine its tendency.

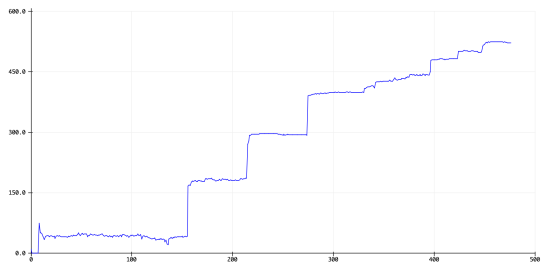

Fig. 9: Data of the sensor over the time

Fig. 9 shows that at a first stage the data increments are approximately similar to one other, suggesting proportionality to the applied pressure. However, after some coins it seems that the pressure is not completely applied to the sensor surface, limiting the increments. Moreover, it has to be mentioned that to make possible the data acquisition, it is needed a threshold pressure that has to be passed out in order to make the sensor able to measure, in this case this happens using a coin of one euro (7,5g) and two coins of twenty cents (12g); it can be established that this threshold it's $\sim 20g$. This design is not precise enough to make a correlation between the output value and the pressure applied because not all the coins are reflected on the data. Another experiment more precise to establish a numerical relationship has to be designed.

2. *Experimental design 2*

To ensure that all the pressure is entirely applied to the sensor surface, the sensor surfaces are covered by a silicone. In order to pass the threshold pressure, the sensor is now between the arms of a clamp. With this experimental design, knowing that the output is straightly proportional to the pressure applied, will be determined the maximum output value to make a linear correlation:

Min. value	1 unit	Increment
Max. value	1024 units	1023 units
Min. Force	$\sim 20g$ or $0,2N$	Increment
Max. Force	$\sim 200g$ or $2N$	$\sim 180g$ or $1,8N$

Table 2: Parameters used to make the correlation

Using that the surface of the sensor is $1,7e\text{-}4\ m^2$, and that $P = F/S$ (where F is the force applied and S is the surface of the sensor), it is found that:

$$P(V_{out}) = k * X_{out} \tag{5}$$

where X_{out} is the output value. Using a linear fit:

$$k = \frac{\Delta X_{out}}{\Delta P} = \frac{1023uts.}{0,180Kg * 9,8m/s^2/1,7e-4m^2} \tag{6}$$

solving this: $k = 9,57Pa/uts.$

$$P(X_{out}) = 9,57 * X_{out} \tag{7}$$

finally the correlation between the pressure applied and the output is found, the Eq. 9 will be useful to compare the simulations and the data acquisition of the experimental results.

B. Experimental results

In this experimental procedure will be used the sensors to study the pressure in the points of Fig. 2 to determinate the shape of the turbulence following the hypothesis in the Fig. 1. The study will include the sphere and the cylinder as the regular forms; and an egg shaped object to see if the pressure pattern is similar to the sphere's one. The experimental results are presented in Figs. 10-12 where the left figure is the data acquisition of the pressure and the right is a modeling of the YZ plane.

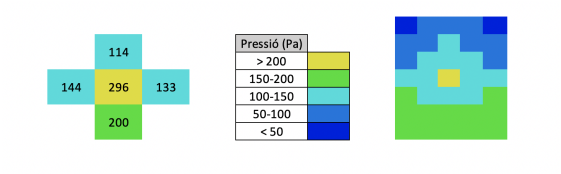

Fig. 10: Experimental results for a sphere.

Where there is the pressure calculated using Eq. 7 and a modeling of the X plane as the simulations figures (see Fig. 6).

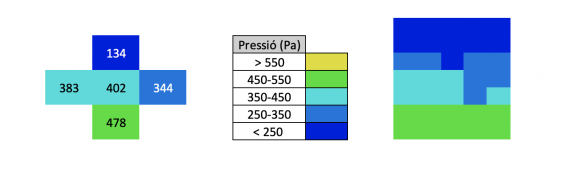

Fig. 11: Experimental results for a cylinder.

Where the pressure is calculated by Eq. 7 for a cylinder and a modeling of the X plane as the simulations figures (see Fig. 7).

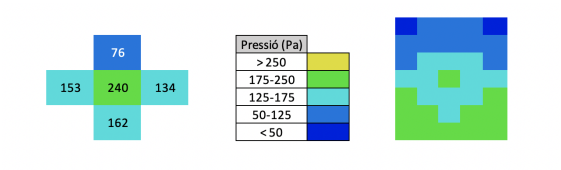

Fig. 12: Experimental results for an egg shaped object.

Where the pressure is calculated by Eq. 7 for an egg shaped object. It is shown in the modeling that the shape of the turbulence is quite similar as in the sphere figure (Fig. 10).

V. DISCUSSION

As the results show, the modeling of the data acquired in all the objects that have been used for the experiment are quite similar to the simulations figures. Also, It can be seen that the data acquired for the egg shaped object is very similar to the data of the sphere, this results leads to an optimal approximation for the irregular object to the regular one. Each figure has been measured five times in order to make an average of the data and minimize the error associated to all the measurements, at the end the results and the modelings are the results of this statistical procedures.

VI. CONCLUSIONS

- It has been developed a successful experimental procedure used to acquire pressure data in order to create a database.
- The hypothesis of the Fig. 1 that the pressure is different inside the turbulence zone is correct, as the results show. It can be seen in all the modelings that the pressures are quite similar to the simulations.
- The simulations validates the results so it can be used this experimental method to determine the optimal approximation of an irregular object to a regular one in order to take the correct parameters to solve Eq. 2 and then use this results to calculate h using Eq. 3 for simplify the solving of heat transfer problems in which convection is involved.

VII. FUTURE STEPS

Improving the system by adding more sensors in order to get more information of the pressure behind the object and being able to get a model with a better resolution.

Also creating a big database with more regular objects will be useful to get better approximations of the object being studied. Also it can be registered different sized objects in order to be able to do an interpolation with the data measured and get an optimal approximation.

Acknowledgments

I would like to thank my advisor Dr. Anna Vilà Arbonès for the help and guidance in this project. Also I would like to thank in a very special way to my father who has helped more than in an economical way, I wouldn't be writting this without him.

[1] Frank Incropera. *Fundamentos de transferencia de calor*. Prentice Hall, 4th ed.

[2] Joel Gonzalez Marroquin. *Determinación experimental de coeficientes de transferencia de calor para convección libre y forzada*, Nicolas de los Garza, N.L. 1998.

[3] Camil Rodrígez and Arturo Hernández. *Sistema para la medición de deformación usando galgas extensiométricas metálicas*, RIELAC, Vol. XXXVII 1/2016.

[4] Brian W. Evans. *Arduino programming notebook*, August 2007.

[5] J. N. Reddy. *The finite element method in heat transfer and fluid dynamics*. CRC Press, 3rd ed.

[6] SimScale. Simulation software online. http://www.simscale.com