TREBALL FINAL DE GRAU

TÍTOL DEL TFG: Experimental analysis for peristalsis drag reduction technique

TITULACIÓ: Grau en Enginyeria d’Aeroports

AUTOR: Raül Rubio Serrano

DIRECTOR: David Pino

DATA: 21/02/16
Overview

The aim of the project is to check experimentally whether the simulations concerning the peristalsis drag reduction technique are valid.

The work has two main parts: the first 4 chapters is an attempt to locate peristalsis inside the turbulence world, beginning from the most general and basic description, then one of the most important analysis: the channel flow, followed by a briefly description of coherent motions inside the flow and finally a description of the flow control given peristalsis in a flow control technique. The second part describes how is designed the actuator, the components, the electric system and the measurement devices. Finally the results are presented.

To create the peristalsis wave has been chosen an actuator based on the shape memory alloys. 204 actuators contracts the silicon pipe forming a wave that is not sinusoidal but it a step discretization of it.

The key parameter to compute drag reduction is the pressure gradient on the active section, a decrease on the pressure gradient means the technique is working. Results obtained show this behaviour but due to its high uncertainty it is not possible to affirm we have achieved our goal. Other reason for why the results are not valid is because the actuators’ displacement is less than the designed one and this fact on the simulations leads to a not-effective effect in drag reduction. To solve the displacement problem there is proposal to install a lunge that absorbs the extra pressure induced by the reduction in volume when the silicon pipe is deformed.
# INDEX

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>MOTIVATION AND OBJECTIVES</td>
<td>1</td>
</tr>
<tr>
<td>INTRODUCTION</td>
<td>2</td>
</tr>
<tr>
<td>CHAPTER 1. TURBULENCE</td>
<td>3</td>
</tr>
<tr>
<td>CHAPTER 2. CHANNEL FLOW</td>
<td>5</td>
</tr>
<tr>
<td>2.1. Near Wall shear stress</td>
<td>7</td>
</tr>
<tr>
<td>2.2. Velocity profiles</td>
<td>8</td>
</tr>
<tr>
<td>2.3. The friction law</td>
<td>13</td>
</tr>
<tr>
<td>2.4. Reynolds stresses</td>
<td>14</td>
</tr>
<tr>
<td>2.5. Pipe flow</td>
<td>17</td>
</tr>
<tr>
<td>CHAPTER 3. COHERENT MOTIONS</td>
<td>19</td>
</tr>
<tr>
<td>3.1. Models</td>
<td>20</td>
</tr>
<tr>
<td>CHAPTER 4. FLOW CONTROL</td>
<td>23</td>
</tr>
<tr>
<td>4.1. Drag reduction</td>
<td>24</td>
</tr>
<tr>
<td>4.1.1 Active techniques</td>
<td>24</td>
</tr>
<tr>
<td>CHAPTER 5. ACTUATOR DESING</td>
<td>27</td>
</tr>
<tr>
<td>CHAPTER 6. THE SYSTEM</td>
<td>31</td>
</tr>
<tr>
<td>6.1. Components</td>
<td>33</td>
</tr>
<tr>
<td>6.1.1 Filtering element</td>
<td>33</td>
</tr>
<tr>
<td>6.1.2 Honeycomb</td>
<td>34</td>
</tr>
<tr>
<td>6.1.3 Piezometer duct</td>
<td>34</td>
</tr>
<tr>
<td>6.1.4 Breathe valves</td>
<td>35</td>
</tr>
<tr>
<td>6.1.5 Hydraulic pump</td>
<td>36</td>
</tr>
<tr>
<td>6.1.6 Active section</td>
<td>37</td>
</tr>
<tr>
<td>CHAPTER 7. ELECTRIC SYSTEM</td>
<td>38</td>
</tr>
<tr>
<td>7.1. Energy supply</td>
<td>38</td>
</tr>
<tr>
<td>7.2. Measurement devices</td>
<td>40</td>
</tr>
<tr>
<td>CHAPTER 8. RESULTS</td>
<td>41</td>
</tr>
<tr>
<td>CONCLUSIONS</td>
<td>44</td>
</tr>
<tr>
<td>ACKNOWLEDGMENT</td>
<td>45</td>
</tr>
<tr>
<td>BIBLIOGRAPHY</td>
<td>46</td>
</tr>
</tbody>
</table>
MOTIVATION AND OBJECTIVES

The project has been done at Politecnico di Milano in the aerospace department with a research group formed by the professors Maurizio Quadrio, Marco Belan, Franco Auteri, Elena Borlandelli and Marco Morandini.

The motivation of the chosen work is my concerning fluid mechanics and the attractiveness to deal with an open research topic. The expectation in the arrival was learn from theoretical meaning the mean concepts of turbulence combined by the laboratory work.

The question tried to solve is whether experimentally peristalsis can achieve the amount of drag reduction showed in the simulations. As the budged available was quite low the technology used is an actuation based in shape memory alloys to form the travelling wall deformation wave known as peristalsis.

The work done before my arrival was the design wave’s parameters (amplitude, frequency, speed) using direct numerical simulation. After that the design and characterization of the actuators and the instrumental equipment needed for the operation to reach the waves’ target parameters.

Once I arrived the system blew up thus two new goals come out, the first one fix all the broken parts and prepare the system for new tests and the second one analyze the results obtained in the previous proves.
INTRODUCTION

The study of turbulence is one of the most important ones concerning fluid mechanics. What is turbulence is still an open question thus there is no definition of it, just a description of its main characteristics. Due to its apparently chaotic nature the analysis of turbulence was a statistical description of the flow. Now, with the development of simulation techniques, are seen some coherent structures inside the flow that are responsible for turbulence production.

As engineers the control of the flow is the principle purpose. Many techniques have been developed with different aims: lift increase, drag reduction or transition control.

Nowadays drag reduction is one of the issues on the industry. Almost the 100% of fluid flows are turbulent, the problem with that kind of flows is the higher energy costs needed to its transport (the flow through a duct), or the movement of objects inside it (airplanes) compared to laminar flows. The cause of this increase in the energy needed is the contribution to the skin-friction drag of extra terms called Reynolds stresses that appear when flows become turbulent.

Thanks to numerical simulations is possible to check new methods for the reduction of drag. Some studies concerns passive technique such as riblets [3] and active techniques as spanwise oscillating wall [5] or spanwise travelling waves [6]. The results obtained with oscillating wall or deformation wall aroused the research on active techniques and in 2011 was done the first simulation concerning peristalsis [8]. This travelling deformation wall technique achieved great drag reduction results.

The data obtained in the simulations must be validated experimentally and then try to convert it in an application for the industry. In this way some experimental implants have been done like in [9]. Following this work, born the present one with the aim of validate experimentally the amount of drag reduction obtained on the simulations with peristalsis technique. Peristalsis consists in a wall deformation travelling wave modulated in space and time that has reached 40% of drag reduction in numerical simulations. In the laboratory of Politecnico di Milano has been built a system consisting in a silicone pipe with a fully developed turbulent flow passing inside it and actuators based in shape memory alloys that deform the pipe obtaining the wave.
CHAPTER 1. TURBULENCE

This chapter is a briefly description the main characteristics of turbulence and three types of it

Most of the phenomena occurring in the day basis involve turbulent flows:
- Blood flow in vessels
- Combustion processes
- Water in rivers
- Boundary layer over plane wings

Even its importance there is no a clear definition about what turbulence is, but all turbulent flows have remarkable properties:

- Random process
  - Time and space dependent
  - The flow statistics are reproducible

- No scale separation
  Whether in laminar flows there is a strong scale separation, in turbulent flows there is a continuum separation of scales
  - Large scales:
    - Small viscous effects
    - Dictated by the geometry
  - Small scales:
    - Large viscous effects
    - Dictated by the viscosity

- Small scale random vorticity

- High Re is needed
  - Turbulence is generated by instabilities in laminar flow at a high Re

- Turbulence is dissipative
  - Energy is dissipated in the small scales
  - Energy is supplied in the large scales
  - Energy dissipation has the timescales of the small scales
  - Exists an energy cascade throughout the scales.

- Is a continuous phenomenon
  - Small scales are much larger than the mean free path
  - The continuum approximation holds, which implies flow governed by Navier-Stokes equations

- Turbulence is 3D
- **Convective flow**: convection process is characteristic for turbulent flows contrary to what happens in laminar flows thus it is structured in layers and the predominant phenomena is the diffusion.

There many theories about what is turbulence and if there are kinds of it or it is a unique concept, but the most accepted one talks about the existence of three types:

**Homogeneous isotropic turbulence**
Homogeneous turbulence is rarely found in flows of practical relevance. Nevertheless, homogeneous turbulent flows have interest in turbulence research because they permit to study selected turbulence interactions isolated from others. In addition making possible to look in more clear way turbulence dynamics, also simplifies the mathematical description and the solution of the equations. Furthermore, homogeneity in space enables the use of periodic boundary conditions, which, in turn, allow studying the turbulence dynamics in a fraction of actual flow space, making these flows very attractive for direct numerical simulations (DNS). Finally, flow homogeneity makes easier experimental set up and enables the turbulence phenomena to be studied in well controlled conditions.

**Flows with mean shear**
Free shear flows are inhomogeneous flows with mean velocity gradients that develop in the absence of boundaries.

**Wall flows**
Wall flows are the turbulent ones produced in the presence of solid walls, for our purpose, check the simulation from Nakanishi et al [8] are the most important ones, thus the only wall turbulent flow treated.

Beginning from the most general case the channel flow, passing through the coherent motions in turbulent flows and drag reduction, the first 4 chapters are an attempt to situate peristalsis technique inside the word turbulence.
CHAPTER 2. CHANNEL FLOW

Channel flow is the basic geometry concerning wall flows and the results are still valid for pipe flow, which is the chosen geometry for our experiment. The analysis done enables us to know the velocity profile and, as water is a Newtonian fluid, derive the expression for the stress. Is also important knowing how Reynolds stresses are distributed to focus in which region drag reduction techniques must actuate.

Is the flow passing through a rectangular section duct. The duct (see figure 2.0.1) has height \( 2\delta \), a long aspect ratio \( (b/\delta >>1) \) and is long \( (L/\delta >>1) \) making possible a predominant flow in the axial\((x)\) direction and a dependence in the velocity just in the cross-stream\((Y)\) direction. In the spanwise direction \((z)\) the dimension is large compared to \( \delta \), and then the flow becomes statistically independent in this direction far from the walls.

![Figure 2.0.1. Channel flow scheme [1]](image)

The analysis done focuses its attention on the central part of the duct, far from the entrance where is located the developing region. Hence in the studied part the flow becomes fully developed and is statistically stationary and statistically one dimensional, just varying statistics in \( Y \) direction.

Important parameters:
- Length scale \( \delta \)
- Velocity scale
  \[
  U_b = \frac{1}{2\delta} \int_0^{2\delta} <U> dy
  \]
  Where \( <U> \) is the average velocity on \( x \) direction
- Reynolds number
  \[
  Re_b = \frac{U_b \delta}{\nu}
  \]
  Where \( \nu \) is the dynamics viscosity
Under the previous the continuity equation simplifies in:

\[
\frac{d < V >}{dy} = 0
\]

The term \(d<w>/dz\) is 0 since \(w\) is 0 and the term \(d<u>/dx\) cancels because the velocity \(u\) is independent of \(x\).

The y-momentum equation reduces to

\[
0 = - \frac{d < v^2 >}{dy} - \frac{1}{\rho} \frac{\partial < p >}{\partial y}
\]

Integration the previous equation with the boundary condition \(<v^2>_{y=0}\) the result is:

\[
v^2 + \frac{p}{\rho} = \frac{p_w}{\rho}
\]

Where \(p_w\) is the pressure at the bottom wall. A significant result in deduced from the previous equation if we look at the x direction gradient, as \(v^2\) is independent of \(x\), we obtain the following:

\[
\frac{\partial < p >}{\partial x} = \frac{\partial p_w}{\partial x}
\]

What has an important meaning, the pressure gradient is uniform along the mean flow direction.

While the x-momentum equation remains:

\[
0 = \nu \frac{d^2 < U >}{dy^2} - \frac{d < uv >}{dy} - \frac{1}{\rho} \frac{\partial < p >}{\partial x}
\]

Being \(uv\) the resulting product when introducing the Reynolds decomposition \((U = <U> + u)\), in the general momentum equation.

As the fluid is Newtonian is possible to establish a relation between the shear stress and gradient velocity components, such as:

\[
\tau = \rho \nu \frac{d < U >}{dy} - \rho < uv >
\]

Hence

\[
\frac{d\tau}{dy} = \frac{dp_w}{dx}
\]
As \( \tau \) is a function of \( y \) and \( p_w \) is a function of \( x \) hence \( d\tau/dy \) and \( dp_w/dx \) must be constant.

Thanks to the antisymmetric about the mid-plane character of \( \tau \) is possible to write both the previous equation and \( \tau(y) \) in terms of the wall shear stress:

\[
\tau_w \equiv \tau(0); \quad \tau(2\delta) = -\tau_w
\]

The solution is:

\[
- \frac{dp_w}{dx} = \frac{y}{\delta}
\]

And

\[
\tau(y) = \tau_w \left(1 - \frac{y}{\delta}\right)
\]

### 2.1. Near Wall shear stress

The shear stress has two different terms, one due to the friction and Reynolds stress:

\[
\tau = \rho \nu \frac{d < U >}{dy} - \rho < u v >
\]

Friction  Reynolds

The behavior of both terms is clearly different and can be seen plotting them as a function of \( y \) as in figure 2.1

![Viscous and Reynolds shear stress profiles](image)

**Fig 2.1. Viscous and Reynolds shear stress profiles [1]**

In one side the friction term is governed by the viscosity hence is relevant close to the wall while the Reynolds stresses are relevant throughout the whole channel.
To analyze the near wall phenomena we have to define the appropriate lengthscales and velocity scales:

- Friction velocity

\[ u_\tau = \sqrt{\frac{\tau_w}{\rho}} \]

- Viscous lengths scale

\[ \delta_v = \nu \sqrt{\frac{\rho}{\tau_w}} = \frac{\nu}{u_\tau} \]

- Reynolds number

\[ Re_\tau = \frac{u_\tau \delta}{\nu} = \frac{\delta}{\delta_v} \]

The distance from the wall is measured in wall units:

\[ y^+ = \frac{y}{\delta_v} = \frac{u_\tau y}{\nu} \]

Measuring in wall units is possible to define the separation of scales, there are two main of them:

- \( Y^+ > 50 \rightarrow \) outer layer: no direct effect of viscosity on shear stresses
- \( Y^+ < 50 \rightarrow \) viscous layer: direct effect of viscosity on shear stresses
- \( Y^+ < 5 \rightarrow \) viscous stresses overwhelm shear stresses. It is not a main region but it has crucial effects on drag production

### 2.2. Velocity profiles

The analysis done to compute the velocity profiles in the different regions is based only in dimensional considerations.

Since channel flow is fully determined by \( \rho, \delta, \nu \) and \( u_\tau \) is possible to derive a formula for the dynamic magnitude \( \frac{d<U>}{dy} \) which is the one that determines the stress and the turbulence production. It depends just on two non-dimensional groups:

\[ \frac{d<U>}{dy} = \frac{u_\tau}{\nu} \Phi \left( \frac{y}{\delta_v}, \frac{y}{\delta} \right) \]

Where \( \Phi \) is a non-dimensional function.
According to Kolmogorov’s hypotheses:

- If Re is large enough, an inner layer exists where $<U>$ is determined by viscous scales alone
- If Re is large enough, an outer layer exists where $<U>$ is determined by outer scales alone
- If Re is very large, an intermediate layer exists where $<U>$ depends on the viscous length alone

The different layer can be seen in figure 2.2.1

![Fig 2.2.1. Scheme of layers in a turbulent channel flow [1]](image)

**Inner layer**

At high Reynold number an inner region exists close to the wall ($y/\delta << 1$) where the velocity profile is just dependent on the viscous scales, so the function $\Phi$ tends to be a function just of $y/\delta_v$

$$\frac{d <U>}{dy} = \frac{u_\tau}{y} \Phi_1\left(\frac{y}{\delta_v}\right)$$

Where:

$$\Phi_1\left(\frac{y}{\delta_v}\right) = \lim_{\delta \to 0} \Phi\left(\frac{y}{\delta_v}, \frac{y}{\delta}\right)$$

At this point is interesting to switch variables to wall unit variables, the resulting equation is the following:

$$\frac{d <u^+>}{dy^+} = \frac{1}{y^+} \Phi_1(y^+)$$

Where:
$u^+ = \frac{U}{u_\tau}$

So the integral is the law of the wall:

$$f_w(y^+) = \int_0^{y^+} \frac{1}{y'} \Phi_1(y') dy'$$

The important is not the equation itself, but the fact that the profile just depends on $y^+$ and, after experimental verification, the fact that $f_w$ is universal.

Universality in two senses:
- Independent of the geometry
- Independent of the fluid properties

The experiments demonstrated a linear profile in the whole viscous sublayer $y^+ < 5$ (see figure 2.2.2)

![Near wall velocity profile](image)

**Fig 2.2.2. Near wall velocity profile [1]**

**The outer region**
Far from the wall ($y^+ > 50$) there is the region in which the viscosity effects are completely insignificant, it means the function $\Phi$ tend to be a function just of $y/\delta$.

$$\Phi_0 \left( \frac{y}{\delta_v} \right) = \lim_{\delta_v \to \infty} \Phi \left( \frac{y}{\delta_v}, \frac{y}{\delta} \right)$$

**The velocity defect law**
The difference between the centerline velocity $U_c$ and the mean velocity profile $U$ is known as the velocity defect law.

Analogue to the inner layer case:

$$\frac{d < U >}{u_\tau} = \frac{dy}{y} \Phi_0 \left( \frac{y}{\delta} \right)$$
Integrating from $y/\delta$ and 1, we obtain:

$$\frac{U_c - U}{u_\tau} = \int_\frac{y}{\delta}^1 \frac{1}{y'} \Phi_O(y') \, dy' = F_D(y/\delta)$$

Contrary to the inner case the experiments show not universality in $F_D$, just universal for the channel flow. It makes sense because in the large scales the influence of the geometry is crucial for the flow behavior.

Increasing even more Re appears an overlap region between the inner and the outer regions, where both laws apply.

Thus:

$$\frac{y \, d < U >}{u_\tau \, dy} = \Phi_I \left( \frac{y}{\delta_p} \right) = \Phi_O \left( \frac{y}{\delta} \right)$$

The only way to satisfy the previous equation is both $\Phi_I$ and $\Phi_O$ being constant:

$$\frac{y \, d < U >}{u_\tau \, dy} = \frac{1}{k}$$

Integrating we obtain another logarithmic law

$$\frac{U_c - U}{u_\tau} = -\frac{1}{k} \ln \left( \frac{y}{\delta} \right) + A_1$$

Where $A_1$ is a constant with value 0.2

The accuracy of the previous formula is shown in figure 2.2.3 where the defect law is compared to DNS results.

Fig 2.2.3, Mean defect velocity profile. Dashed line log law; solid line DNS results [1]
The log law
When Re is high enough there is an intermediate region that follows a logarithmic law on the mean velocity profile. It occurs per $y^+ > 50$ and $y/\delta < 0.1$, here the viscosity effect vanishes since we are far from the wall but $y/\delta$ is too small for the outer variables to matter. It implies a constant value for the function $\Phi_1$:

$$\Phi_1(y^+) = \frac{1}{k}$$

Where $k$ is the Von Karman constant with value 0.41.

Thus the velocity profile:

$$\frac{d < u^+ >}{dy^+} = \frac{1}{y^+ k}$$

Integrating we obtain

$$u^+ = \frac{1}{k} \ln(y^+) + A$$

A is a constant, the experimental testing showed a value of 5.2

Figure 2.2.4 shows the experimental validation of the analytical results obtained

Fig 2.2.4. Mean velocity profiles in fully developed turbulent channel flow measurements [1]
2.3. The friction law
Defining the velocity profiles enable us to find the relation between the Reynold number and the skin friction coefficient.

A good estimate for the bulk velocity \(U_b\) is obtained by using the log law through the whole channel since in the outer layer the error is really low and, even in the inner layer there is a big error, the region is so small that the contribution to the integral is insignificant.

The inner layer follows the following expression:

\[u^+ = \frac{1}{k} \ln(y^+) + A\]

While in the outer layer:

\[\frac{U_c - U}{u_\tau} = -\frac{1}{k} \ln \left(\frac{y}{\delta}\right) + A_1\]

Adding both

\[\frac{U_c}{u_\tau} = \frac{1}{k} \ln \left(\frac{\delta}{\delta_v}\right) + A + A_1\]

What is equal to:

\[\frac{1}{\sqrt{c_f}} = \frac{1}{k} \ln(Re_c \sqrt{c_f}) + A + A_1\]

The experimental validation of the previous result is seen in figure 2.3

*Fig 2.3. Skin friction coefficient against Reynolds number; Dashed line laminar case \(c_f = 16/3Re\); solid line turbulent log law equation [1]*
2.4. Reynolds stresses

Reynolds stresses are the cause of an increase in drag in turbulent flows compared to laminar flows. These stresses are produced by fluctuations in the velocity components so \( u, v, w \). Understanding how they are produced could lead to mechanisms that prevent these fluctuations to act, thus, reducing the drag.

In figure 2.4.1 can be seen the velocity fluctuations distribution:

![Graph showing velocity fluctuations](image)

Fig 2.4.1. \( u, v, w \) as a function of \( y/\delta \) [1]

The previous figure shows clearly the anisotropy of the flow and different intensity of turbulent phenomena depending on the distance from the wall.

There are three different behaviors what leads into three regions to analyze separately. In the log-law region the normalized stresses (\( \frac{u_i u_j}{k} \)) are uniform as the rate of production-dissipation rate \( P/\varepsilon \) and the mean shear rate \( \frac{S_k}{\varepsilon} \).

Near to the centerline the velocity gradient and the shear stress vanish, so \( P \) tends to 0. This behavior is seen in figure 2.4.2

![Graph showing Reynolds stresses and turbulent kinetic energy](image)

Fig 2.4.2. Reynolds stresses and turbulent kinetic energy normalized by the friction velocity against \( y^+ \) [1]
Near the wall (see figure 2.4.3) occur the strongest turbulent processes: production, dissipation and turbulent kinetic energy reach the maximum value per $y^+$ less than 20.

At this point it is interesting to develop in Taylor series the velocity components near the wall.

$$u_i = a_i + b_i y + c_i y^2 ...$$

All the $a_i$ terms are zero for no-slip or no penetration condition.

For continuity the term $b_2$ is 0:

$$\frac{\partial v}{\partial y} = 0$$

Then the Reynolds stresses are the energy associated to the velocity components fluctuations. They grow from the wall in the following way:

$$u^2 = b_1^2 y^2 + ...$$

$$v^2 = c_2^2 y^4 + ...$$

$$w^2 = b_3^2 y^2 + ...$$

$$uv = b_1 c_2 y^3 + ...$$

DNS simulations (see figure 2.4.4) confirmed the analytical expression for the Reynolds stresses.
Experimental study for peristalsis drag reduction technique

Fig 2.4.4. Kinetic energy and Reynolds stresses on the viscous sublayer [1]

Where \( k \) is the mean value for the main velocity fluctuation components

\[
k = \frac{u^2 + v^2 + w^2}{3}
\]

Defined turbulent kinetic energy is possible to derive an equation for it which in this geometry reduces to:

\[
\frac{d}{dy}\left(\frac{1}{2}\left(\langle vu \cdot u \rangle + \frac{\rho' v'}{\rho} \right) - \nu \frac{d}{dy}(k + \langle v^2 \rangle)\right) = P - \varepsilon
\]

The interesting point is look how each one of this terms evolve as a function of \( y^* \), what is shown in the next figure 2.4.5

Fig 2.4.5. \( P_k = \) production; \( D_k = \) viscous diffusion; \( T_k = \) turbulent transport; \( \varepsilon_k = \) dissipation rate; \( \pi_k = \) pressure transport. [1]
The production term grows from the wall as \( y^3 \) and reaches its peak at \( y^+ \) around 12, here the rate \( P/\epsilon = 1.8 \) and this extra energy is transported away. At the wall peaks the energy dissipation which is balanced by the diffusion term whereas the pressure transport is negligible for the whole channel.

### 2.5. Pipe flow

This section is located here because of the similarity in the results with channel flow. Pipe flow (see figure 2.5.1) is the only one that can be simulated and experimentally tested, here lies its interest.

The relevant scales for this geometry are:
- Lengthscale: Diameter \( D \)
- Velocity scale:
  \[
  U_b = \frac{1}{\pi R^2} \int_0^R 2\pi r dr
  \]
- Reynolds number
  \[
  Re = \frac{U_b D}{\nu}
  \]

The law of the wall holds:

![Velocity profiles for a fully developed pipe flow](image)
Each curve is done at different Reynolds number, as it increases the logarithmic region extends.

As happens with the channel flow geometry the velocity profile is linear in the inner layer and follows the function $f_w$ which is universal. Then appears the logarithmic region but there is a little difference, the constant $k$ takes a value of 0.44 while the constant $A$ takes a value of 6.1. See figure 2.5.2

In this case for measurement simplicity the friction factor is expressed in terms of the pressure drop on the axial distance.

$$f = \frac{\Delta p}{\frac{1}{2} \rho U_B^2 L}$$

Again integrating the logarithmic velocity profile, Prandlt’s friction law is derived:

$$\frac{1}{\sqrt{f}} = 2 \log(\sqrt{f} Re_b) - 0.8$$

The law is well verified in the experiments as show figure 2.5.3:

![Graph showing friction factor against Reynolds number](image)
CHAPTER 3. COHERENT MOTIONS

This section talks about the influence of coherent motions in the flow. It is thought they are responsible for turbulent production so cutting them with some mechanism could lead in a relaminarization of the flow. DNS results show the presence of these structures and, simulating the effect of peristalsis with this code, is possible to design the fundamental parameters for our actuation technique.

In turbulent flows the kinetic energy from the mean flow is converted into turbulent fluctuations and dissipated into internal energy by the viscous action. The production and dissipation phenomena have been studied for a long time but the complexity of turbulence inhibits great results.

Most statistical description ignores the presence of quasi-periodic coherent patterns throughout the flow which are apparently the responsible of the sustaining turbulence. Depending on the region we look at we can find different phenomena:

- The viscous sublayer and buffer region are organized into alternating narrow streaks of high and low-speed fluid that are persistent. The most turbulence production occurs in the buffer layer during intermittent ejections of low speed fluid and incoming high-speed rushes. This process is believed to be quasi-cyclic and is called bursting, but there is no consensus about the continuity of the cycle.
- The inner region appears to be self-sustaining but today is believed that the outer structures have an influence on the phenomena near the wall which is Re dependent.

What is observed in turbulent boundary layer depends on the tools used, so a set of coherent structures has been reported. These structures have been organized in eight types but the most relevant are the next four ones

- Low-speed streaks
- Ejections of low-speed fluid outward from the wall
- Incoming high-speed rushes towards the wall from the outer region
- Vortical structures

In the last years the development of numerical techniques has allowed the chances to know how these coherent structures are formed and how they interact with the flow.

Two main simulations have been used on the investigation of coherent structures: large eddy simulations (LES) and direct numerical simulation DNS. In LES the smallest scales are modeled while the large ones are computed using Navier-Stokes equations. DNS compute using Navier-Stokes equations for all scales. They are needed because its accuracy and the not enough spatial resolution in LES to look at the near-wall features. The problem with DNS is the high computational cost it has.
One of the concepts that has evolved with the use of simulations is term known as bursting, thought to be the responsible for turbulence production. Most of the definitions of bursting describe strong events intermittent in time but numerical investigations have revealed an alternative picture: the way that turbulence is produced is not burst-like and seems to be more intermittent in space than in time. DNS results at low Reynolds number also show near-wall turbulence is dominated by quasi-streamwise vortices which eject low speed fluid away from the wall. There is also the presence of arches and transverse vortices but in minor number compared to quasi-streamwise vortices in the buffer region, where the bursting process seems to originate.

The study of vortices is one of the challenges since they seem to be the cause of turbulence for two reasons:
- In a boundary layer, any vortex with an orientation other than wall normal has the potential to function as a pump that transports mass and momentum [2]
- There is no a mathematical definition of vortex.

3.1. Models
Conceptual models are proposed to describe the behavior of turbulent boundary layers.

The first one was proposed by Theodorsen in 1952 who described a hairpin vortex model for turbulence production and dissipation using visual techniques. These kinds of techniques were focused on the buffer layer and reported a maximum production on it and the importance of the coherent motions concerning that production.

In this model vertical “tornadoes” form near-wall regions of low-velocity fluid and grow outwards with heads inclined downstream at 45°, and with spanwise dimension proportional to the distance from the wall [2]. The graphical description is in figure 3.1.1.

Fig 3.1.1. Primary structure of wall-bound turbulence [2]
The progress in informatics technology enabled to process data in digital computers what changed the focus to the bursting and its scaling. The complete test model during this period comes out from Smith (1984). What he proposed applies for the wall region ($y^+ < 100$) and describe the kinematics and dynamics of hairpin vortices.

The model says that the “bursting” of a low-speed streak is the probe of vortex roll-up formed on the top and sides of the streak. Once formed, a vortex loop moves outward by self-induction and downstream owing to the streamwise velocity gradient. The trailing legs of the loop remain in the near-wall region but are stretched, forming quasi-streamwise vortices that serve to pump fluid away from the wall and to accumulate low-speed fluid between the legs. Coalescence of stretched legs of multiple hairpins is postulated as a mechanism by which low-speed streaks are preserved or redeveloped [2] (see figure 3.1.2).

![Streamwise Vortex Stretching](image)

Fig 3.1.2. Illustration of the breakdown and formation of hairpin vortices during streak-bursting process [2]

Finally DNS methods made possible a detailed dynamical analysis for all the flow variables, bursting and wall cycle study.

Robinson in 1990 proposed a model for low-Reynolds number. In this model, quasi-streamwise vortices dominate the buffer region, while archlike vortices are the most common structure in the wake region. In the logarithmic, or overlap, region, both arches and quasi-streamwise vortices exist [2]. The representation of it is in figures 3.1.3 and 3.1.4.
Fig 3.1.3. Idealized schematic of vertical-structure in different regions of the turbulent boundary layer [2]

Fig 3.3.4. Conceptual model of kinematical relationships between (1) ejecton/sweep motion and quasi-streamwise vortices in the near-wall region and (2) Arch-shaped vortical structures in the outer region [2]
CHAPTER 4. FLOW CONTROL

Peristalsis is a flow control technique but there are many of them and with different objectives. This section tries to locate peristalsis inside the flow control concept and at the end there is the explanation of it.

Flow control involves all those techniques trying to influence the flow to take benefits from its properties.

All the techniques can be enclosed in the following scheme:

![Flow control scheme](image)

In passive techniques some features of the system are changed (material, shape...) in order to obtain a different behavior of the flow while in actives techniques actuators are introduced in order to intervene in dynamics of the system.

Inside the active ones there two groups:
- Predetermined (Open loop): here the actuation is previously though and it does always the same action
- Reactive(Closed loop): here the system is able to readjust the actuation action as a function of the real-time parameters

There are three main flow control objectives:
- Transition control
- Separation control
- Turbulence control
That could lead into diverse technological benefits such as:

- Drag reduction
- Lift increase
- Mixing increase
- Noise reduction

### 4.1. Drag reduction

Drag coefficient for fully developed two dimensional channel flow is computed by the formula:

\[
C_f = \frac{12}{Re_b} + 24 \int_0^1 (1 - y)(-uv)dy
\]

The first term correspond to the laminar case while the second one is due the Reynold stresses. Has been demonstrated that the minimum drag occurs in laminar flows, and this minimum is an absolute one, it means the drag cannot be reduce below the laminar case, so the ultimate aim of the techniques is reach the relaminarization of the flow.

Passive actuations regarding drag reduction try to prevent turbulence development. A significant example is the use of riblets: they are grooves on wall surface oriented in the flow direction that could reach 10% the drag reduction [3]. Problems related with riblets are the weakness against the pollution particles decreasing its efficiency.

Other methods consist in the introduction of particles in the fluid such as polymers that have the capacity to change the fluid viscosity.

Although all the applications for aeronautics must become passive, we will focus on the actives ones such peristalsis is an active technique

#### 4.1.1 Active techniques

Today most of the ongoing research about the drag reduction concerning the contact between the fluid and the wall is done in open loop, the idea is to actuate in the turbulent production phenomena cutting them.

One of the key parameters choosing the actuation is the forcing direction due to the high anisotropy in near-wall region. Forcing along the streamwise direction is usually thought to be a less effective means to affect the flow. On intuitive grounds, forcing in the wall-normal direction is usually thought of as the ‘best’ way of forcing, but it disrupts the natural state of turbulence significantly, at the finite amplitudes typically required by an open-loop control. Finally, forcing in the spanwise direction is found to be quite effective, and this work addresses several open-loop techniques based on this method [4].
**Spanwise oscillating wall**

Is the first attempt to achieve significant results. The idea was that a sudden spanwise pressure gradient could reduce temporarily turbulent skin-friction. Making the spanwise pressure periodic in time could lead into high drag reduction.

DNS study combining pressure gradient and wall deformation what lead into the simplest form of wall oscillation [5]

\[ w = A\sin(\omega t) \]

Where A is the amplitude and t is the actuation period \( T = \frac{2\pi}{\omega} \)

With such actuation the wall cycle is partially suppressed reporting a drag reduction up to 40%.

**Spanwise travelling waves**

The simulation made in [6] the flow was forced with volume forces following the next expression:

\[ F_z = Ie^{-\frac{z}{\Delta}} \sin(k_z z - \omega t) \]

Where I is the intensity that decreases exponentially away from the wall, \( k_z \) is the wavelength and t time.

Drag reduction of 30% are measured for low Reynolds number.

To see in a clearer way the effect of the wave parameters in [7] was transformed the previous wave into a space-time distribution of wall acceleration.

The wall velocity follows the next formula:

\[ w = A\sin(k_z z - \omega t) \]

There are registered drag reduction also of 30% but the problem it the energy supplied for the actuation is higher than the savings in terms of drag.

**Streamwise travelling waves**

The idea is born to see whether or not is useful to do two spanwise in terms of drag reduction. Quadrio et al [8] have studied this effect using the following equation:

\[ w = A\sin(k_x x - \omega t) \]

where the forcing still acts along z, but is modulated along x, so that the waves may travel in the streamwise direction with phase speed \( c = w/k_x \) [4]

The DNS obtained results are plotted in a complicated map in which the drag reduction decrease or even increase depending on the parameters.

The maximum drag reduction achieved is 48%
Peristalsis consists in a wall deformation wave in the vertical plane which travels downstream (see Figure 4.4.1.1) and modulated in time and space.

The wall deformation follows the following expression:

\[ y(x, t) = A \sin(kx - \omega t) \]

Where \( A \) is the semi-amplitude, \( k \) is the wave number, \( \omega \) the frequency, \( x \) the current coordinate and \( t \) the time.

The study done in [8] considered an \( A \) comparable to the thickness of the viscous layer and not also demonstrated that drag is reduced but under some parameters the flow is relaminarized in the whole channel.

The results obtained in the simulations [8] are showed in figure 4.1.1.2:

The important case is number 11 where the pressure gradient almost reaches the laminar case.

The drag reduction rate \( R_D \) is defined as:

\[ R_D = \frac{P_0 - P}{P_0} \times 100 \]
Where \( P = -\frac{dP}{dx} \) is the current gradient pressure and \( P_0 \) is the pressure gradient for the uncontrolled flow.

And the net power saving is defined as:

\[
S = \frac{W_p - (W_p + W_a)}{W_p} \times 100
\]

where \( W_p \) and \( W_a \) are the pumping power and power required for wall deformation and \( W_p \) is the pumping power for uncontrolled flow.

**CHAPTER 5. ACTUATOR DESIGN**

*This chapter is a basic description of how was designed the actuator in the work done by Politecnico’s research group in [10]*

The system has similarities with the experiments described in [9], in the same way it has been chosen a circular pipe independently the plane channel study in the simulation. This is because in such conditions as large enough diameter the results are still valid, pipe flow is the only feasible geometry and the computational cost is reduced with plane geometry.

The technology chosen for contracting the pipe is shape memory NiTi wires due to its simple method for contract and expand them with heat.

Nitinol are alloys composed by 55% Ni and 45% Ti. The characteristic phase transform for shape memory is a thermoelastic martensitic transformation between two solid phases, austenite and martensite, that is stable in a determined temperature range. This temperature range is delimited by conventional values that indicate the onset and offset of the direct transition from austenite to martensite during cooling (martensite start \( M_s \) and finish \( M_f \)), and of the reverse transformation during which austenite is created from martensite (austenite start \( A_s \) and finish \( A_f \)) [10]. Figure 5.0.1 shows the phase transform:

![Phase transform](image)

Fig 5.0.1. Phase transform [11]
and figure 5.0.2 shows the diagram length-temperature:

![Figure 5.0.2. Temperature - length diagram [11]](image)

Thus applying heat or cold enables to produce a phase transforms what leads into an increase or decrease of the wire’s length.

As the wire must be heated and cooled cannot be directly attached to the silicone pipe thus a support must be inserted in between. This fact is crucial for the system as the wires’ length is one of the more restrictive parameters thus the contraction on the pipe applied depends on it. But no just the tension, the heating process is triggered by electric current so more length means more material to heat or cold influencing directly on the duty cycle. In figure 5.0.3 in defined the duty cycle:

![Figure 5.0.3. Duty cycle [10]](image)

The other parameter to be determined is the wires’ diameter: bigger diameters enable bigger contractions, longer wires’ life (more working cycles) but more power supply is needed and the cooling phase is decayed.
From the numerical study did, it have been chosen a deformation wave with the following parameters (see figure 5.0.4):

- Amplitude $\Delta \eta^* = 7$
- Period $T^* = 17.5$
- Phase speed $c^* = 28.3$

The simulation done at $Re_T = 180$

![Sketch illustrating the geometry of a pipe flow with peristaltic waves.](image)

Fig 5.0.4. Sketch illustrating the geometry of a pipe flow with peristaltic waves. Each wall point experiences a displacement $\Delta \eta(x, t) = \eta_{max} \sin(2\pi \lambda (x - ct))$.

[10]

Once known the wave is time to choose wires’ dimensions, the challenge as discussed before is the balance between a fast enough cooling phase without decaying the heating phase. The best fitting wires’ cooling method has been studied in several experiments [12, 13]. Finally the adopted one is discussed in [14] where the contact between NiTi and aluminum achieve good results for our purpose.

A NiTi element which works cyclically in tensile configuration under a typical stress can recover up to 4% of deformation, providing a constant, stable and reliable contraction regardless of the number of cycles.

Our application has never been experimentally investigated, but on the basis of the two available numerical studies, it can be estimated that the best-performing peristaltic waves are achieved with a pipe contraction of 8%, which is our design target. In order to reach that contraction by using a NiTi wire working at a percentage lower than 4%, we need to design a rigid device that is able to geometrically amplify the wire linear contraction [10].

The wires cannot be attached to the pipe because temperature reached by them can damage it and the cooling phase would be not effective enough so aluminum disks are installed in between. As the electric current it is desired to pass just through the wires an isolating layer is put. Figure 5.0.5 shows the whole disk actuator
Fig. 5.0.5 (a) A variation $\Delta L$ of the length of the circumference $L_0$ results in a variation $\Delta L \pi$ of the diameter $D_0$. (b) Section of pipe and actuator, showing the device supporting the SMA wire. The pipe has internal diameter $D_i$ and thickness $t$. The wire support has height $h$, so that the NiTi wire length is $\pi (D_i + 2t + 2h)$. (c) Cross-section of the grooved interface between the SMA wire and the actuator. A thin Kapton layer (60 \mu m) is placed between aluminum and wire to provide electrical insulation. Groove diameter is chosen to maximize the contact surface between Kapton and the wire [10].

During every cycle a contraction is produced by heating and the following expansion because of the internal pressure of the fluid.

The stress applied by the wires is computed using the next formula:

$$\sigma = \frac{pD_i z}{\pi D_w^2 z^2}$$

Where $D_w$ is the wire diameter, $p$ the internal pressure and $z$ is the width of the device out of its plane.

With:

$p= 0.4 \text{ bar} \quad D_w= 0.2 \text{ mm} \quad z= 8\text{ mm}$

A stress of 200 MPa is obtained which is the ideal one for this kind of material.
CHAPTER 6. THE SYSTEM

The experiment blew up at the beginning of the works so it changes the provided schedule. The system was dismounted and cleaned, there was corrosion impurities inside it, substituted the broken wires and the aluminum pipes for anodized ones. This section contains a general description of the whole system and specifically the most important components without the electric part that is treated in the next chapter.

The system is a hydraulic circuit driven by a pump. Due to the pressurization condition needed it is a closed system. There are two branches (see figure 6.0.3) divided by the active section. The first one compresses the part between the pump and the entrance of the active section and the second one is the responsible of bringing the fluid back to the pump.

In the first branch are located the flow measurement and the piezometric duct which is a fundamental component as enable to adjust the static pressure inside the system. The working pressure is decided to be 0.3 bar and should be carefully calibrated as the system is very sensible to pressure changes. Figure 6.0.1 shows schematically the components of the system.

Fig 6.0.1. System scheme [14]
The active part (sezione di prova) is where the actuators are located forming the deformation wave, there are 204 actuators. Parallel to the active section two wood supports bed the cabling responsible for the actuators' performance (see figure 6.0.2).

Fig 6.0.2. Active section and cabling

Fig 6.0.3. At the left the first branch, at right the second branch; the green tubes are attached to the pump

At the beginning and at the end of the action section there are two holes connected to the pressure transducer that gives us the friction factor measurement.

The system is also provided with a thermistor, a laser that measures the actuators displacement and an amperemeter to know the current through the wires.
Data acquisition is done by a PXI connected to a computer. Also through the PXI is possible to control the actuation performance changing the duty cycle.

The wave created is on purely sinusoidal, the sinus is discretized by a step function. How it is created is in the following figure:

![Discretization of the peristalsis wave](image)

6.1. Components
For a good operation of the system a set of components have been installed, these ones are described in the following section.

6.1.1. Filtering element
The working fluid is water and it is introduced in the system by the hydraulic network of the lab what could bring some undesired particles so must be filtered. The filter (figure 6.1.1) is made of metals that can actuate also as a disinfectant.

![Filtering elements](image)
6.1.2. Honeycomb
The function of this component is to break eventually air bubbles going round the system and attenuate possible velocity peaks produced by the pump. Another positive effect of the honeycomb is the orientation of mean flow velocity component in the axial direction killing the vorticity generated by the pump. It is made by aluminum and put just after the pump, being part of the first branch discussed before. The figure 6.1.2 shows the honeycomb installed in the connector.

![Fig 6.1.2. Honeycomb](image)

6.1.3. Piezometer duct
Its function is to introduce and maintain the working pressure, also permits to know the pressure variation.

The duct has a manometer at the end (see figure 6.1.3.2), just before the manometer there is tap communicated with the atmospheric pressure. When the tap is closed there is air remaining inside tube, this air actuates as a hydraulic spring. When the system is running there is a volume decrease in the active section due to the non-purely sinusoidal wave, then the spring helps to regulate this extra pressure without decaying the actuators performance. Another vantage of the duct is its capacity to trap air bubbles.

The duct is provided with a scale graduated in millimeters for its calibration, show in the next figure.

![Fig 6.1.3.1. Pressure reference](image)
The manometer scale is in bar with a resolution of 0.5 bar and a range between -1 bar and 1.5 bar.

To the pressure lecture on the manometer must be added the water column pressure to compute the working pressure.

The static pressure is:

\[ p = \rho g h \]

As the working fluid is water it is equivalent to:

\[ 1\text{cm} = 0.001 \text{bar} \]

Thus, controlling the rate between water and air inside the duct is possible to fix the desired pressure in the system. For the experiments is has been decided 0.3 bar.

![Fig 6.1.3.2. Piezometer duct at the left; manometer at the right](image)

### 6.1.4. Breathe valves

It is crucial to remove the air bubbles trapped on the fluid, to optimize this process are installed these devices. Unlike what happened in the piezometric duct, where the bubbles are trapped but remains inside it, in this case the air is removed from the system.

Breathe valves are little plastic taps and there are two installed, one in the first branch and another one in the second, are located in a higher position (see figure 6.1.4) given that the air tends to rise up as it is less dense than water.
Fig 6.1.4. Breather valve

6.1.5. Hydraulic pump

Is the responsible for the fluid motion and has a maximum angular velocity of 1500 rev/min. The pump is powered with 380 V. The rotating regime is controlled by an inverter which is controlled at the same time with Labview software. For a good operation the pump must have the impeller immersed, and then is located in a lower location respect to the circuit branches and the power cord is buried to prevent some errors in the measurements.

Fig 6.1.5. Pump connected with the two branches
6.1.6. Active section

The action section is formed by a silicone pipe where the peristalsis waves have to achieve the relaminarization of the flow.

To generate the wave are set 204 disk actuators, each one of them is formed by 4 aluminum sectors as support of the NiTi wires. When the wires are heated the disks squeeze the silicone pipe generating the wave. All the actuators are located between four aluminum plates; these ones are the support for the alignment bars.

The whole active section beds in wood supports.

Previous components are seen in figure 6.1.6.1

![Alignment bars, wood supports, disk actuators with visible NiTi wires and one of the pressure intakes.](image)

Fig 6.1.6.1.; Alignment bars, wood supports, disk actuators with visible NiTi wires and one of the pressure intakes.

**Disk actuators**

As discussed in the previous paragraph its main function is the contraction of the silicon pipe, but they have other extra ones:

- **Heat dissipation**: the contraction is produced by heat excitation of the wires by electric current. The cooling phase as discussed in chapter 4 is one of the crucial processes, to improve the cooling the disks are made of aluminum. As is not desirable the current passing through the aluminum an isolating layer made of Kapton is added.

- **Expansion**: when the heating phase finishes the actuator must return to its initial position. The extra pressure inside the pipe is transferred to the wires by the aluminum pieces
The separators of the actuators and the working squeezing scheme are seen in figure 6.1.6.2

Fig 6.1.6.2. Silicon pipe with lower disc mounted and separators at the left; operating scheme at the right.

CHAPTER 7. ELECTRIC SYSTEM

This chapter explains how the system is governed and how the data is acquired. The power plant blew up at the beginning without a clear reason so another one with the same characteristics was bought.

7.1. Energy supply

The power plant has been built by an external company CELM S.n.c. For the experiment 204 wires have to be excited with a resistance of 11.8 Ω. To form the peristalsis wave the 204 disk actuators are divided segments of 6, making a periodic actuation of 34 segments. The actuation is done in a way in which the wires activated occupy the same position in each segment, it means, the first actuators of each segment receive electric current simultaneously and so on with the other 5 ones.

The power plant is plugged to the 380V lab mains, transformed in direct current by a diode bridge that charges a battery. The battery is installed to achieve the peak current demand for 34 actuators.

The requirements are the following:
- Power peak: 22542W (7.5A * 88.4V * 34 wires)
- Mean power: 5728W
- Frequency: from 0.5 Hz to 5 Hz

The power plant works with 6 plates (see figure 7.1.2) with mosfet switches, each plate is able to supply current to 34 wires and is controlled by a function generator. Each plate has 12 clamps with 12 cables; each cable is divided in 3 when it arrives at the wood support (see 7.1.1).
This configuration enables us the control of every single actuation line and the management of possible isolated failures. All ground connections are done in a way in which do not disturb measurements.
7.2. Measurement devices

Our goal is to check whether the drag reduction is achieved and the amount of it, which, as discussed in chapter 4 is computed by $R_D$ parameter

$$R_D = \frac{P_0 - P}{P_0} \times 100$$

So the crucial measurement is the $\Delta p$ along the active section, to do that is installed a differential pressure sensor (see figure 7.2). It has a range between -2mbar and +2mbar with a sensitivity ±0.1% of the full scale.

The transducer is set in parallel to the active section with four intakes, two upwards it and the other ones downwards.

![Image of pressure transducer]

Fig 7.2. Pressure transducer

In both cases (upwards and downwards) the intake points are located in the same longitudinal position but two of them are in bottom of the aluminum tube and the other two at the top. To control from which points we are measuring an electro valve is inserted in order two open or close the corresponding ducts.

Capillarity effects could disturb the results so the ducts colligated to the transducer have a diameter of 6mm and a constant section in all of them length but in the intake (hole done in the aluminum pipes) where the section is reduced. That is because the intake holes have to be really small in order to attenuate as much as possible them effect in the flow.

Temperature is measured by a thermistor PT100 supplied with 24V. PT100 is a thermistor based in current, it means, the output is a range between 4 and 20 mA depending on the temperature. It converted in an output between 0 and 10V for data acquisition requirements.

Flow rate is measured by a flowmeter with an output range between 2 and 10V, 2V indicate 0 flow rate. The calibration for this instrument has remained the same as in [9] for the precision of the operation.
CHAPTER 8. RESULTS

As we are trying to check what Nakanishi et al [8] did on the simulations the important measurement is the pressure gradient.

Many tests have been done; the last one is seen in the next figure:

Fig 8.0.1. Head losses against time. Working frequency 2Hz at Re 4900

The result obtained is the expected one, in the first 1000 unit times the pressure gradient decrease, after this point the system stabilizes and the pressure gradient remains constant in average.

In the firsts moment it was though the obtained results were correct but the high uncertainty on the measurements unable us to from a clear judgment. In a first attempt to clarify whether are valid or not, we changed the inputs of the pressure transducer but unexpectedly the pressure difference still positive in other words the same figure 8.0.1

Trying to explain the previous fact it is interesting to take a look to the actuator displacement thus is the cause of the relaminarization of the fluid and the simulations showed high dependence on this parameter. Plotting the laser data against time (see figure 8.0.2) is clearly seen the actuators don’t move enough:
Fig 8.0.2. Actuators displacement against time; green curve corresponds to the test with 1 actuator, orange one is the test with 6 actuators and purple one is the test with the whole system running.

As the number of actuators increase there is a reduction of them displacement. This is produced because of a pressure increase inside the active section. When just 1 wire is actuating the pipe is able to manage the local area reduction and the effect is negligible but when the whole system is running there is a bigger volume reduction. This volume reduction increases the pressure making the actuators unable to contract enough. Can also have a negative effect the friction between two consecutive actuators. Another problem reported is a missing volume when the whole system is running. Looking at the piezometer duct the height water increase is 5 cm what means 40 cm³ and computing the volume change with the displacement (assuming linear profile between two consecutive actuators) the result is 144 cm³. The fact can be explained by the non-total circularity of the pipe, it has elliptical shape what makes the contraction less efficient but it is just a theory. Another possible explanation is the non-contraction of the 4 actuator segments it is to say the contraction in two of them leads in an expansion of the other ones, but those two theories have not been confirmed.

The last one is the presence of impurities in the water. The corrosion of the aluminum has two effects, the first one is the presence of particles in the fluid and the second one is the deformed shape of the internal pipes’ wall disturbing the flow. New anodized ones have been bought to solve this.
In a first attempt to solve the actuators displacement problem it is planned a test with a half of actuators running, in order to avoid the friction and enable a higher deformation of the pipe. In the case the attempt doesn't work there is the proposal for the installation of a lung that absorbs the extra pressure.

There are two possible options for doing that (see figures 8.0.3 and 8.0.4):

Install it at the end of the piezometer duct

![Fig 8.0.3. Lung installed at the end of the piezometer duct](image)

The vantage of this configuration its technological simplicity but there are doubts about the effect of pressure waves through this segment and the effectiveness because the piezometer is already installed to enable possible expansions.

The other one is locate two lungs, one at the beginning and at the end of the active section:

![Fig 8.0.4. Lunges installed at the beginning and the end of the active section](image)

The vantage of this proposal is the constant pressure in the active section what seem to be crucial for the good system operation. In the other hand is complicated because the holes in the aluminum pipes have to be really small to do not disturb the flow.
CONCLUSIONS

The present work had the aim to check experimentally the results obtained on the simulations using peristalsis technique and the objective has not been reached. In the first tests the results obtained have too much uncertainty to affirm the system is working.

After few tests the system blew up so it changed the entire expected schedule changing the goal to repair it and analyze the cause of the obtained results in the tests. The first part is almost done: broken wires are replaced, a new power plant has been installed and the problem with the corrosion of the aluminum is solved. Looking at the obtained results in the previous tests we realized they were wrong for two reasons:

- To check the good operation we did a test changing the intakes of the differential pressure sensor what would have lead into a negative sign of the results but unexpectedly we obtained the same curve.
- The other symptom is seen looking at the actuators displacement. The design displacement is 2.7 mm when the system achieve stationary regime but the laser show 0.3 mm

The displacement problem is thought to be the cause of the bad operation thus the simulations showed a high sensitivity on the results on it, changing the displacement could lead into a 0 or even an increase of drag. The reason for why the actuators do not move enough lies in the pressure increase when the system is running. As the system is closed and water incompressible under the experimental conditions, the volume reduction of the pipe increase the working pressure and this extra working pressure enables the actuators to contract enough. To solve the problem comes out the idea of installing a lung that absorbs this extra pressure.

The future work to be done is check all the measurement instruments and the design of the lung. The lung design seems to be a challenge thus there are two proposals to install it and the stiffness of it is crucial.

Summarizing even if the primary goal has not been achieved, the new ones that are fixing the damaged parts and the analysis of the results are reached: the system is practically prepared for running (just leaving assemble the new pipes and reconnect the system to the pump) and concerning the results a proposal for solving the bad operation have come out.
ACKNOWLEDGMENT

My gratitude specially to professor Maurizio Quadrio for give me the opportunity to work with him and all his support, and to the other components of the team Marco Belan, Elena Borlandelli, Franco Auteri and Marco Morandini.

I also would like to thank professors Luigi Quartapelle, Aldo Frezzotti and Luigi Vigevano for the lunchtimes did together and to my Italian teacher Alessandra Battioni for being something else than a teacher.

Thanks to my family and my friends for supporting me in the hard moments and celebrating with me the good ones.

And finally express my gratitude to professor David Pino for his readiness, patience and support.
BIBLIOGRAPHY


