Heveas problem: The Fountain

Adnan Balci and Maria Bruna and Chris Budd and Dongdo He and Poul G. Hjorth and Grzegorz Krzyżanowski and Krystian Kubala and Christian Lund and Berit Martensen and Gitte Schmidt and Monika Sikora and Damian Smug

28. August 2013

Heveas problem: The Fountain

Group contributors:

Adnan Balci (DTU), Maria Bruna (U. Oxford), Chris Budd (U. Bath),
Dongdo He (Hong Kong U.), Poul G. Hjorth (DTU), Grzegorz Krzyzanowski,
Krystian Kubala (U. Wrocław), Christian Lund (KU), Berit Martensen (DTU),
Gitte Schmidt (DTU), Monika Sikora (U. Wrocław), Damian Smug (U. Wrocław).

Figure 1:

1 Introduction

HEVEAS ApS is a small Danish company with a range of innovative products for the home and office. Company webpage: www.heveas.com.

1.1 Problem description

HEVEAS asked the study group to explore the operating and design characteristics for a small (cm scale) device that, when inserted into the vertical (often turbulent) flow from a water mains faucet, can generate a smaller, vertical jet (as in a drinking fountain). See the figure:

Figure 2: The basic principle of the device. The photo is a still from a movie supplied to us by Heveas.

1.2 Design requirements

Several things are crucial to make this device function in a satisfactory manner:

- The exit jet should ideally be a smooth laminar jet for easy drinking. Enough flux must come through so that one can drink quickly (annoying when you have to wait 10 seconds to fill in your mouth with water).
- Robustness: Operating the device should not require delicate 'tuning' of directions and positions when inserted into the faucet stream. Also, the entry flow should not create a huge amount of splashing.

1.3 Questions:

Specific questions for the group to consider are:

- What is the basic fluid mechanical principles governing this device?
- Does the height and flux of the output fountain depend in a simple way the properties of the incoming water flux?
- Is there an optimal duct shape? (what should be the purpose of the duct)?
- What is the optimal material? one that is rough, so that we lose energy from friction (if theory is go from turbulent to laminar), or rather one that is very smooth and almost no energy losses?
- What would be the effect of adding baffles into the internal chamber? reduction of the effective lengthscale (from increase of wetted perimeter), i.e. making flow have lower Reynolds number, or actually helping pressure chamber stay 'pressurised'?

1.4 Background physics

Here we assemble a few key relations, formulae, and definitions regarding fluids, drawn from a number of sources, e.g., Cimbala and Cengel [1] (book with online sample chapter, with e.g. the garden hose example) or White [18], Turbulent to laminar transition: [13], Friction effects: classic paper [11], and [9, 13].

Hydrostatic pressure. A fluid of density ρ at rest in a vessel in a gravitational field g has a (hydrostatic) pressure p_{hyd} which varies with depth z as

$$p_{hyd}(z) = \rho g z$$

Bernouilli's Principle.Along each streamline in a flow with no free boundaries, the following quantity is constant (Bernouilli's Principle):

$$\frac{p}{\rho g} + z + \frac{v^2}{2g} = \text{constant}$$

Reynolds number. For a flow of a liquid of density ρ, viscosity μ, flowing in a duct with characteristic spatial dimension L and flowing with characteristic velocity U, the dimensionless Reynolds number is:

$$R = \frac{\rho U L}{\mu}$$

The Reynolds number measures in some sense for the flow the ratio between inertial forces and viscious forces. If the Reynolds number is high ($\approx 4 \times 10^3$), the fluid flow becomes turbulent. If the Reynolds flow is much lower, the flow will be laminar.

Froude Number. The Froude number F is a dimensionless measure of the free surface flow fluid velocity U, relative to the surface wave velocity $\sqrt{gh}$:

$$F = \frac{U}{\sqrt{gh}}$$

where g is the acceleration of gravity, and h is the flow depth.

Supercritical, Subcritical. When the fluid has a Froude number which is high ($>> 1$), the flow is laminar, whereas a small Froude number indicates turbulent free surface flow. A famous example of this is the so-called hydraulic jump, where a fast laminar flow slows down and suddenly makes a transition to a slower, deeper, turbulent flow.[reference!]

2 Mathematical models

During the study group we have considered two different approaches to the problem and studied two models, the *kinetic model* (Section 3) and the *pressure model* (Section 4). The former assumes a steady-state flow at nearly atmospheric pressure, so that the device essentially redirects the flow from flowing downwards to upwards. The second model, the pressure model, assumes that the incoming water through the inlet creates a water-jet effect, impacting the reservoir of water already inside the fountain and pressurising it (like a piston). In the next sections we describe each of these two models in detail, and conclude that the second model may provide a better solution to match our design requirements for the device.

3 Kinematic model

One possible design for the fountain is to consider a "curved pipe" such that it takes a continuous (steady) flow from the water source (e.g. the tap) and changes its direction and modifies its properties such that "it is easier to drink from it". Recalling the design requirements, two important properties of the outlet flow are that: (i) it is a stable non-turbulent stream (that is, constant flux laminar flow), and (ii) it is high enough so that we do not have to bend in excess. Regarding the entry point, we would like it to be "splash-free" even thought it is likely that the inlet flow is turbulent.

It is clear from such requirements that there must be a compromise between turning the flow from turbulent to laminar (thus making it diminish its energy) and making the fountain jet as high as possible. In this section we examine this trade-off with a kinematic model.

One mode of kinematic operation is to have a free-surface flow throughout; thus the flow never fills the duct completely (even at the end). The analysis of this mode is simple: The velocity of the flow is simply turned around, with the forces from the duct nearly perpendicular to the streamlines (thus doing no work), and the conditions at the exit with respect to pressure and height z are the same. Thus, with a small deduction due to frictional losses in the duct, we expect the jet to go approximatly to the height of the faucet head (certainly no higher). This may be sufficient. But this mode will waste a lot of water, since the intake must be so small as not to fill the smaller outlet, and the mode is not robust, since a small displacement of the inlet *will* fill the duct completely, and there is a transition to filled-duct kinematic flow.

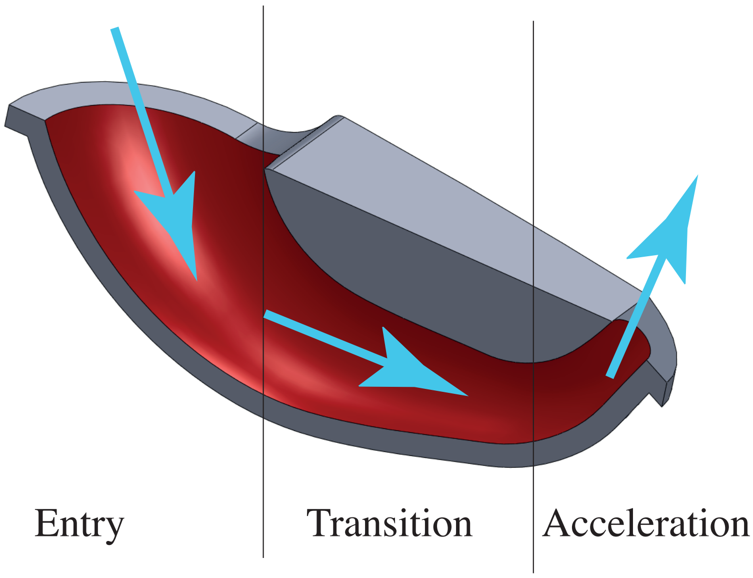

Figure 3: Three kinematic zones.

In the filled-duct kinematic flow we assume that a slight overpressure is generated by the freely falling water being decelerated. By Bernoulli's principle, if the velocity decreases by Δv, the pressure goes up by $\Delta p \approx \rho \Delta v$. This pressure will then drive a flow through the channel. If the flow remains turbulent, the conditions at exit are as those at the entry, but again with frictional losses, thus the turbulent filled-duct flow is not a good device operation either. This is the mode seen in figure ??.

Finally, one may ask: what if the device manages a transition from turbulent flow at the entry to laminar flow at the exit. This is the most interesting case. To explore this, we define three zones as the water travels from inlet to outlet: an entry or funnel, a transition zone (where the flow is laminarised), and an acceleration zone (where the goal is to accelerate the flow by reducing the pipe cross-section). In Figure 3 we show a sketch of a possible design indicating these three zones.

3.1 Entry or funnel

The purpose of the entry zone or funnel is to collect as much water as possible for the inlet jet, and turn it into the pipe region. This must be done in an efficient way (careful design of the entry geometry) so that the least amount of energy is lost in this transition. Typical values of flux from a tap can range between 3 to 18 litres per minute (50-100 ml per second) [16, 17].

The typical Reynolds number at entry is

$$Re = \frac{dU}{\nu} \tag{1}$$

with $\nu \approx 10^{-6}$ m^2/s as the kinematic viscosity of water. The Reynolds number of the tap water jet flow can vary widely, taking values from 200 to order 10^5 [4, 5, 15]. Thus the must take turbulent flow at entry as a possibility (this is in accordance with experience; one can turn the faucet to low, and get laminar flow, and to high, and get turbulent flow).

3.2 Transition zone

The goal of the transition zone is to laminarise the flow, that is, turn it from turbulent to laminar, so that we can have a clean splash-free fountain at the outlet. The premise here is to turn turn the flow laminar but only just, so that it preserves as much kinetic energy as possible. A possible way to do this is to work out the desired Reynolds number for the laminar flow, compare it with the Reynolds number of the turbulent flow at the entry point, and work out how much kinetic energy must be dissipated. Obviously, it is not possible to know with certainty the critical Reynolds number for the transition to occur, and experiments with the specific set-up should be carried out.

In order to make some progress, let's be conservative and suppose that when $Re = 1200$ our flow is laminar, that is, this is our target Reynolds number. The Reynolds number depends on the flow velocity and the characteristic lengthscale. For a perfectly circular and smooth pipe, the latter is identical to the pipe diameter d. A generalised definition of the Reynolds number to non-circular pipes is

$$Re = \frac{d_H U}{\nu}, \qquad \text{with} \qquad d_H = \frac{4A}{C}, \tag{2}$$

where A is the cross-sectional area and C is the *wetted* perimeter. The quantity d_H is known as the hydraulic diameter, and can be thought of as an effective diameter. For a circular smooth pipe, $4A/C \equiv d$, so that Re reduces to the conventional Reynolds number.

From (2) we see an easy way to reduce the Reynolds number (thus laminarising the flow) while keeping the velocity U as high as possible: to reduce the hydraulic diameter d_H. There are two obvious ways to do so:

- Introduce baffles in the pipe, thus increasing the wetted perimeter with a negligible reduction in the cross-section.
- Increase the roughness of the pipe surface.

Both of these increase the wetted perimeter while keeping essentially the area the same as for a smooth pipe of the same diameter (see Figure 4).

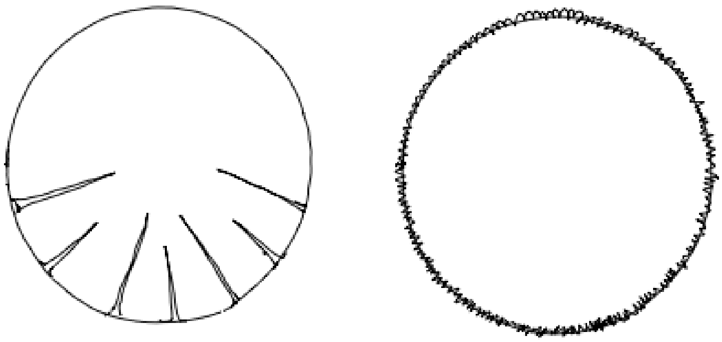

Figure 4: Two design alternatives to reduce the Reynolds number of the flow: baffles (left) and surface roughness (right).

Now we would like to find out the friction losses of the flow in a pipe given a surface roughness and a Reynolds number. For that there exists an experimental chart known as the Moody diagram [11], which gives us the Darcy-Weisback friction factor λ given a relative roughness (defined as the ration of the mean height of roughness of the pipe to the pipe diameter) and Reynolds number [**?**]. For laminar flows (roughly $Re < 2000$), the roughness is negligible and the friction factor can be calculated using $\lambda = 64/Re$ [6]. For example, in [15] they presnet experimental results of the friction factor λ as a function of Re in the whole range of possible tap water Reynolds numbers (and a highly water-repellent wall, so that surface roughness can be neglected). The transition from laminar to turbulent flow is clear in their results (see Figure 5(a) in [15]).

3.3 Accelerator

The accelerator at the exit of the device should speed up the flux whilst preserving as much kinetic energy as possible. Therefore, we suppose the walls of the device are very smooth and that no energy is lost by friction. Then the acceleration of the flow is achieved by a gradual reduction of the cross-sectional area of the pipe, so that by conservation of mass,

$$V_1 = \frac{V_t A_t}{A_1},$$

where V_t and A_t are the velocity and the cross area at the end of the transition zone, and V_1 and A_1 are the velocity and area at the exit point, with $A_1 < A_t$.

If we set the design fountain height to be $h = 10$ cm, then ignoring any friction with air the velocity at the exit point should be

$$V_1 = \sqrt{2gh} \approx 1.4 \text{ m/s}.$$

Now, the standard theory tells us that in general the critical Reynolds number based on the tube diameter above which the flow is turbulent and below which the flow is laminar is $Re_{crit} = 2300$ [10]. Say that to be conservative we would like a Reynolds number of 2000 for a clean water jet at the exit. This implies that the pipe diameter at the end of the accelerator should be $d_1 = 1.4$ mm, which is not a very reasonable number...

4 Pressure model

In this section, we examine an alternative model based on the "water-ballon effect". That is, can we build a device capable of sustaining a pressurised compartment due to the impact from the water jet? If so, is this strategy better at achieving a higher fountain height? A sketch of such device is shown in Figure 5.

First of all, suppose that we had a ballon filled with incompressible flow. Then if pressure is applied over some area of this ballon, because of incompressibility, the pressure will be communicated to the entire chamber. Next imagine that we do a very small hole on the

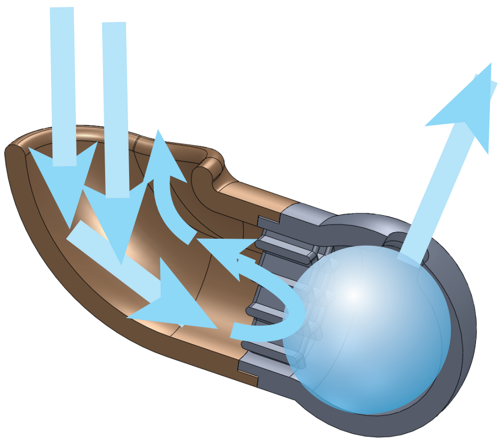

Figure 5: Pressure model geometry. The cavity is filled with water, and the incoming jet pushes on this cavity, creating a pressure, which in turn causes the fountain out of the small exit hole.

ballon's surface, open to the outside, and assume it is small enough so that it only alters the pressure distribution locally. If we ignore the surface tension effects as well as the friction near the exit, and assume a steady flow leaving the ballon, we can use the Bernoulli equation to estimate the height of the water fountain. We know that

$$\frac{P}{\rho g} + \frac{V^2}{2g} + z = \text{constant}. \tag{3}$$

We apply it between the point inside the ballon next to the hole, $z = 0$, and the point at which the water reaches its maximum height, $z = h$. Using that the velocity inside the ballon is relatively low (ignore local effects near hole), we find that

$$h = \frac{P}{\rho g}, \tag{4}$$

where P is the pressure (above atmospheric) inside the chamber.

Add something about the Dimensional analysis? That is, compromise in size: larger devise (less splashing?) but then hydrostatic pressure and viscous forces/friction may overcome jet effect.

4.1 Input flux

Critical to the effective functioning of the device is the input flux and the input velocity, noting that there will be substantial loss of volume at the entry point. Practical experiments show that if the input flux is too low then the device simply will not operate. There are a number of reasons for this. They include: turbulent dissipation, friction loss, air entrainment and the resistance of surface tension [8]. All of these losses are typically associated with *sub-critical flow*. If the input velocity is V_1 and the device diameter is D_1, then an estimate for the Froude Number F_r is given by

$$F_r = \frac{V_1}{\sqrt{gD_1}}.$$

Sub-critical flow occurs if $F_r < 1$ so that $V_1 < \sqrt{gD_1}$. If we take $D_1 = 1$ cm then this gives $V_1 < 0.31$ m/s. This gives an estimate for the lowest speed at which the device is likely to function, although this may be a lower estimate as we have not considered the effects of surface tension or air entrainment fully.

A typical flow rate from a tap (and which we have used in experiment) is 1 m/s.[1] We denote the velocity from the tap by V_0. Then if the device is positioned at a distance h_0 from the tap, the water velocity at the pocket entry is $V_1 = \sqrt{V_0^2 + 2gh_0}$. Taking $h_0 = 5$ cm and $V_0 =$, we then have $V_1 = 1.4$ m/s, which gives a value of $F_r = 4.5$. Interestingly this value is on the transition between steady and oscillating flows, so an input velocity of $V_0 > 1$ m/s or a larger distance h_0 might be best for smooth operation of the device.

4.2 Effect of water jet in the chamber

The water jet will generate a pressure in the liquid on impact, an effect known as the water-hammer effect [2]. This effect has been investigated widely in the context of pressure waves propagating in pipes, which can be a cause of major problems from noise and vibration to pipe collapse. In the first moments after impact, there appears the so-called water-hammer pressure, which only lasts for the short period that it takes for the release wave to be generated [12]. The relevant pressure for us here is the *stagnation pressure*

$$P_{jet} = \frac{1}{2}\rho v_{jet}^2. \tag{5}$$

This can be understood as follows: when the moving column of water is suddenly arrested by a fixed water surface, its kinetic energy (due to a velocity v_{jet}) is transferred into potential energy. (This is assuming that the static fluid is purely incompressible. If compressibility was taken into account, there would be an extra factor in (5)).

[1]This number came from the value of 12 l/minute found on the internet and assuming that the tap has area 2 cm^2. Reference [17] provides a U.S. specification of flow rate of lavatory faucets between 0.8 and 1.5 gallons per minute (gpm), that is, between 3.0 and 5.7 l/min. In Australia, typical taps discharge 15 to 18 litres per minute [16].

4.3 Size of the exit hole

The effectiveness of the device, in particular the shape and size of the fountain, depends upon the size of the exit hole. We assume that the hole is on the top of the device and it has a radius r_2 and an area A_2. In this section we will consider the optimal size of the exit hole.

If the device was flux driven then by flux conservation the exit velocity of the fountain is inversely proportional to A_2 and hence by energy considerations, the fountain height should be proportional to A_2^{-2}. This is certainly not what is observed in practice. In contrast, the model described earlier assumes that the fountain is pressure driven with a pressure P close to the exit hole. In this case, the height h and exit velocity u of the fountain are given by

$$h = \frac{P}{\rho g}, \qquad V_2 = \sqrt{\frac{2P}{\rho}}. \tag{6}$$

Observe that both h and V_2 are *independent of the hole size*. In practice this is not quite what is observed. If the size of the hole is *too small* then the effects of the back pressure constricting the flow, combined with surface tension and viscosity effects act to reduce the exit flow to a negligible amount. In contrast, if the exit hole is *too large* then the pressure in the device reduces, and friction and turbulent dissipation effects also become important. Our intuition is that both h and u will gradually reduce as A_2 increases (see Figure 12 for the numerical results of h versus exit hole size in a cartoon model). This is in good agreement with our lab experiment. From experiments conducted during the Study Group this would seem to be given when $r_2 \approx 1$mm.

There are various other considerations that will affect the choice of the size of the exit hole:

- *Regularity of the fountain:* Smaller values of r_2 lead to laminar flow and a more regular (and higher) fountain. As r_2 is increased the fountain becomes more turbulent and is more likely to break up in the air. This makes it harder to rinse your mouth.
- *Exit flux:* Larger values of r_2 give a greater exit flux. This may be desirable for the optimum rinse (see Figure 12 in the next subsection).
- *Splashing:* The larger the exit flux, the smaller the back flux at the entry of the device. Thus devices with smaller holes will lead to much greater splashing and may be less pleasant to use.

  These considerations indicate that it is worth experimenting with the exit hole size.

4.4 Comsol analysis of the pressure model

In order to test the validity of the pressure model, in this section we consider a much simpler two-dimensional toy model consisting of a circle with two openings. The disk is filled with water (assumed to be purely incompressible) whose motion is described by the Navier-Stokes equations. On the closed walls of the circle, we impose a slip condition (that is, there are no viscous effects at the walls so that no boundary layer develops. Otherwise could also impose a no-slip boundary condition, so that the fluid doesn't move at the wall). At the inlet, the boundary condition must represent the effect the water jet has to the bulk fluid already in the device. We consider two alternative conditions, either prescribing the pressure at the entry (pressure from the jet impact), or a normal velocity condition (assuming that at the boundary the velocity is such of the jet - before the slowing down due to the impact with "still" fluid). At the outlet, we prescribe the open boundary imposing atmospheric pressure and no viscous stress. The geometry of the toy model is shown in Figure 6.

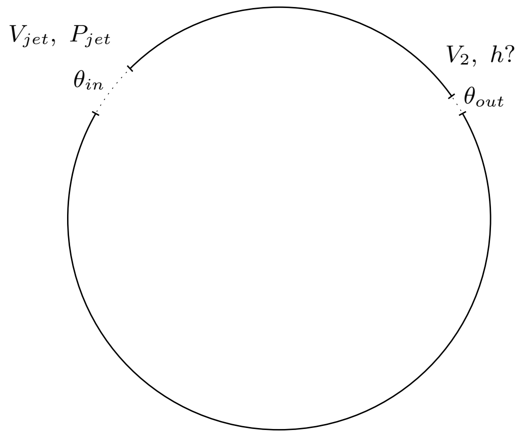

Figure 6: Geometry of the two-dimensional model of the pressure fountain. The inlet has an angle of θ_{in} and the outlet of θ_{out}.

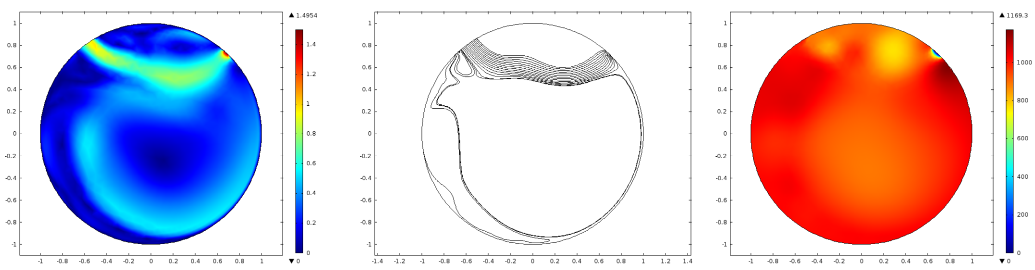

Figure 7: Solution for $\theta_{in} =$ and $\theta_{out} =$. (a) Velocity, (b) Velocity field streamlines, (c) Pressure.

We use the Fluid Mechanics package in Comsol Multiphysics to solve the steady-state problem. In Figure 8 we show results with $\theta_{in} =$ and $\theta_{out} =$ with a pressure condition at the inlet, $P_{in} = P_{jet}$. The plots show the distribution of velocities in the disk, the streamlines of the velocity field (which start at the inlet), and the distribution of pressures.

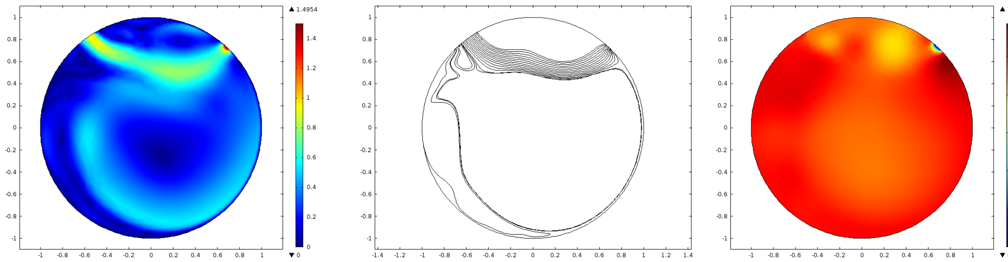

Figure 8: Solution for $\theta_{in} =$ and $\theta_{out} =$ with a pressure condition at the inlet, $P_{in} = P_{jet}$. (a) Velocity, (b) Velocity field streamlines, (c) Pressure.

In Figure 9 we show the same results as in Figure 8 but for the outlet half the arc length, $\theta_{out} =$.

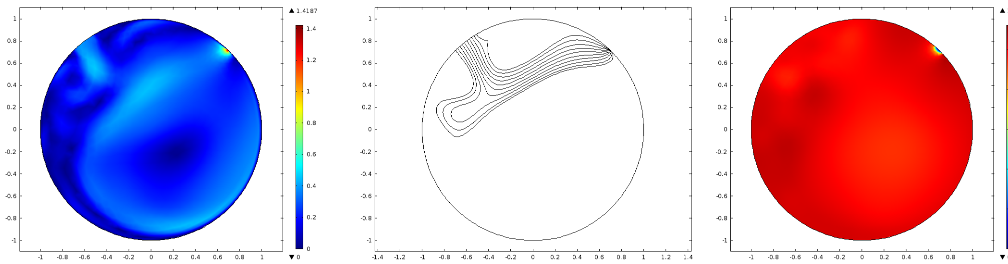

Figure 9: Solution for $\theta_{in} =$ and $\theta_{out} =$ with a pressure condition at the inlet, $P_{in} = P_{jet}$ (the outlet is half the size of that shown in Figure 8). (a) Velocity, (b) Velocity field streamlines, (c) Pressure.

In Figure 10 we show the velocity distribution in the case where instead of prescribing the pressure at the inlet, we impose a value for the normal velocity.

In the next two figures we examine the effect of changing the outlet hole size on the fountain height and flux. For the simulations we choose a ball of radius 1 (nondimensional) fixed and inlet hole size $\theta_{in} = 15°$. We vary the outlet hole size $r_2 = \theta_{out}\pi/180$ for $\theta_{out} = 1, 2, \ldots 11°$ and measure the normal outlet flow speed V_2, the outlet flux $q_2 = V_2 r_2$ and the fountain height h. We find that the outlet velocity is maximal for $r_2 \approx 0.15$ (corresponding to $\theta_{out} = 7° \approx \theta_{in}/2$), see figure 11. It is also for this ratio on outlet to inlet hole sizes that the maximal flux and fountain height are achieved,

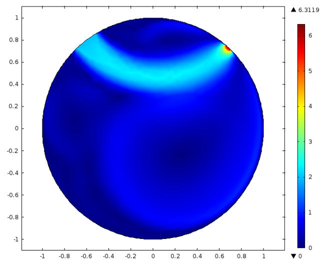

Figure 10: Solution for $\theta_{in} =$ and $\theta_{out} =$ with a normal velocity condition at the inlet, $\boldsymbol{u} \cdot \boldsymbol{n} = U_{jet}$. Velocity distribution.

see Figure 12. Therefore, there is an "optimal" value of r_2 giving the largest fountain exit height.

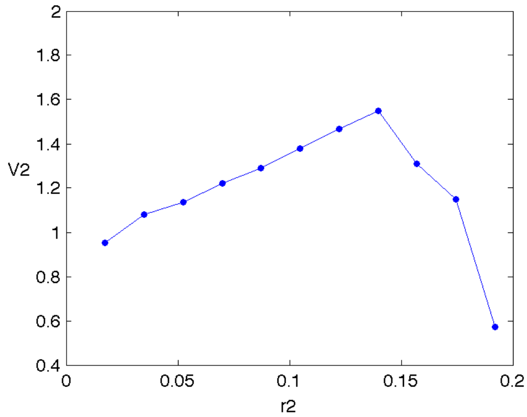

Figure 11: Numerical results for a device with $\theta_{in} = 15^\circ$ and $\theta_{out} = 1, 2, \ldots, 11^\circ$ and nondimensional ball radius 1. Outlet normal velocity v_2 as a function of the exit hole size $r_2 = \theta_{out}\pi/180$.

5 Experiments

5.1 Lab experiments

Laboratory experiments have been carried out continuously throughout the work with the fountain in order to increase our intuition of the different phenomena at work and to test the various model predictions.

The typical experiments took a flexible plastic tube, with a hole cut at one end to allow for entry of water from the tap. The end of the tube was then constricted to give a smaller exit hole. A flux of water from the tap was directed into the tube and

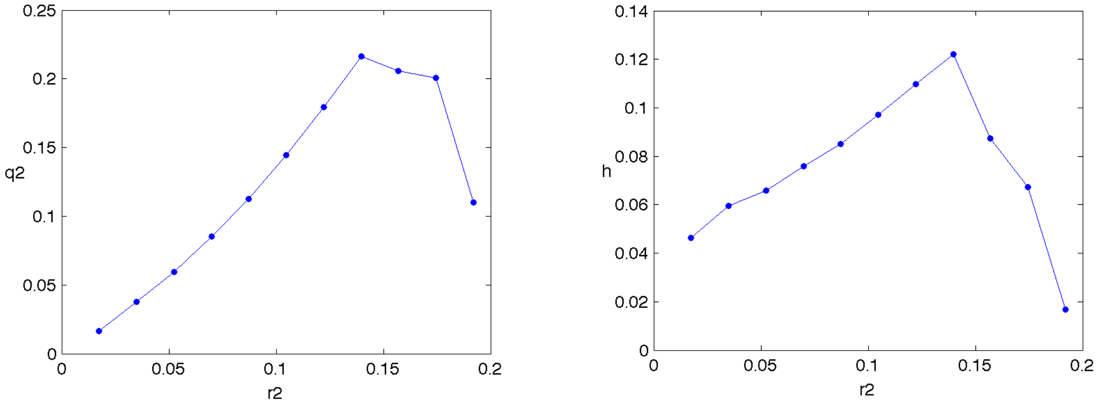

Figure 12: Numerical results for a device with $\theta_{in} = 15^\circ$ and $\theta_{out} = 1, 2, \ldots, 11^\circ$ and nondimensional ball radius 1. Left: Output flux $q_{out} = v_2 r_2$, where v_2 is the average normal flow velocity on the outlet. Right: fountain height h as a function of exit hole size $r_2 = \theta_{out}\pi/180$.

we observed both the fountain height at exit and also the degree of splashing at the exit. Experimental variables that we could control were (i) the rate of flow of water from the tap, (ii) the distance of the device from the tap, (iii) the size of the exit hole, (iv) the size and shape of the entrance hole, (v) the length of the device. The flow rate from the tap could be increased from zero to about 1 m/s.

First experiments confirmed that the distance from the faucet to the device has a substantial impact on the height of the water jet produced. This is consistent with the pressure impulse model. A reasonable estimate of the normal height from faucet to device was determined to be 10 cm, which - combined with looking up standard flow out of modern water faucets - gave us the entry velocities into the device of $1.5 - 2.5$ m/s.

We have experimented with a basic shape of the device as being a straight piece of pipe. In this case where entry and exit hole are the same size, there is a low amount of splash from the entry, but no significant vertical jet can be produced. Here, splash is to be understood as water exiting from the entry point or not entering at all, spreading out from the entry at high velocities. This led us to investigate the impact of reducing the exit hole. Our experiments showed that a small exit produces a laminar jet but more splashing at the entry, whereas larger exit holes produce the opposite, that is, a turbulent exit jet but lower amounts of splash. Furthermore, these experiments suggest that splashing is a significant issue to consider when using the pressure model, since in this case there is always more water coming from the faucet, than can exit the device. Also, if the size of the exit hole is not too big it does not have a big impact on the height of the water jet.

The basic shapes we tested also seemed to indicate that making the entry walls as sharp as possible might reduce splashing. The basic tube device was tested under a number of different entry flow conditions, showing that it can be fairly easy to produce a reasonable jet when using a steady laminar flow. When the flow from the tap is more turbulent, the water jets produced are unsteady, changing height and direction and splashing also gets worse. This only goes to confirm that robustness of the device is a big issue.

Do we have any photos/movies of experiments we can put in the report?

5.2 Comsol simulations

Adnan Balci and Grzegorz Krzyzanowski.

Three dimensional model of flow through a pipe. Include plots of results, as well as model solved, assumptions on boundary conditions, etc.

5.3 Device prototype

The project of the fountain was based on a torus. It is easy to make some modifications of the shape of a model. The first function describes the radius of cross-section. The second function presents the radius of curvature. It is a function of angle. Two examples are shown in Figure 13. It is available to adjust input and output e.g. from circular to ellipsis. All cross-sections perpendicular to the curve are circulars. It is also possible to modify the coarseness of material which is used to make the fountain. The MATLAB code is included in B. The function surf2stl.m was downloaded from Matlab Central. The .stl format is crucial to 3D printing.

3D printing relies on putting a special material (for instant Acrylonitrile Butadiene Styrene) layer after layer. It is a plastic which has a lot of applications in industry e.g. to make the Lego blocks. Printing of this model (about 5cm long) lasts around two hours. After the whole process, the product must be brought under pickling. A picture of the 3D printer we have used in shown in Figure 14.

6 Conclusions and Future work/directions

- Need to be careful when designing a device under the assumption of flux conservation, since as we have seen in experiments there is a lot splashing at the entry point.

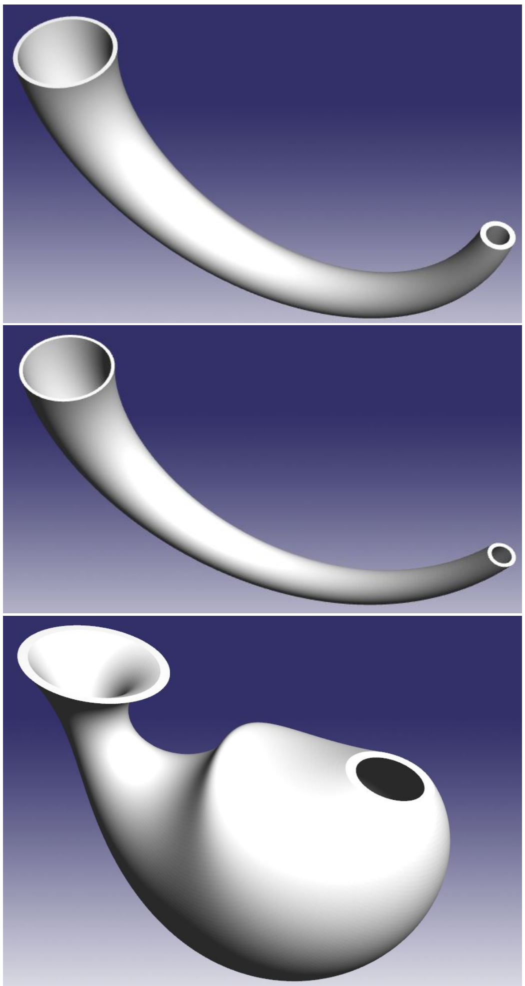

Figure 13: Design examples: linear (first), parabolic (second) and cubic (third).

– The height is less affected by small changes in the size of the exit hole in the pressure model.
– Overall a more stable jet in the pressure model.
– Assumptions in the pressure model seems reasonable.
– The angle of the incoming flow can lead to a significant energy loss.

A Optimal pipe shape

Adnan Balci and Grzegorz Krzyzanowski

In this project, the steady and incompressible newtonian fluid has been considered. With the density ρ and viscosity μ, we have the N.S.E.

$$\rho \mathbf{u}.\nabla \mathbf{u} = -\nabla p + \mu \nabla^2 \mathbf{u} + \rho \mathbf{g} \tag{7}$$

Figure 14: Picture of the 3D printer in the Mads Clausen Institute in Sønderborg.

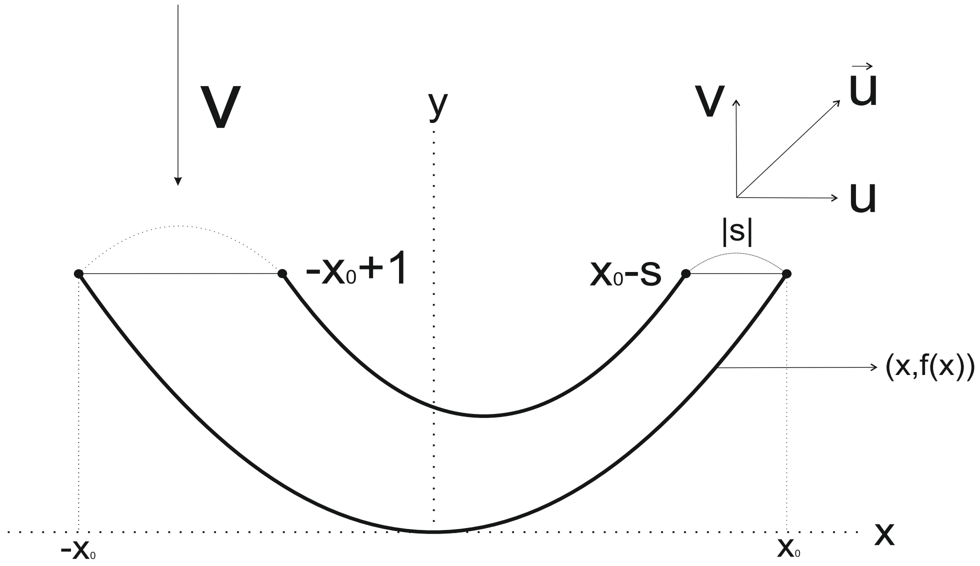

Figure 15: Shape of the project

with the continiuity equation

$$div(\mathbf{u}) = 0 \tag{8}$$

where $\mathbf{g}$ is the gravity, $\mathbf{u} = (u, v)$ is the fluid velocity and p is pressure. Assume that the interface is smooth, we can write $f(x) = kx^2$ and $g(x) = mk(x - \frac{x_0-s}{2})^2 + t$ where a is the parameter and $m, t > 0$ are constants in the region $[-x_0, x_0]$.
A stream function ψ exists from the continuity equation such that the streamlines are found from

$$\dot{x} = u = \frac{\partial \psi}{\partial y}, \quad \dot{y} = v = -\frac{\partial \psi}{\partial x} \tag{9}$$

To obtain local information of the flow closed to a given point that is taken as the origin, ψ is expanded in a Taylor series

$$\psi = \sum_{i+j=0}^{\infty} a_{i,j} x^i y^j \tag{10}$$

We need to the following three conditions:
(a) the kinematic boundary condition at the interface is that the velocity field must be parallel to the interface,that is, $\mathbf{u}.\mathbf{n} = 0$. Since $\mathbf{n} = \frac{(-f(x)',1)}{\sqrt{(f(x)')^2+1}}$ we can say

$$-f^{'}(x)\frac{\partial \psi}{\partial y} = \frac{\partial \psi}{\partial x} \tag{11}$$

insert the expansion (3) and (4) into (5) yields

$$\begin{aligned}
&a_{1,0} = 0, \quad a_{2,0} = k a_{0,1}, \quad a_{2,1} = 2k a_{0,2}, \\
&a_{3,0} = \frac{2k a_{1,1}}{3}, \quad a_{2,2} = 3k a_{0,3}, \quad a_{3,1} = \frac{4k a_{1,2}}{3}, \\
&a_{4,0} = \frac{k a_{2,1}}{2}, \quad a_{2,3} = 4k a_{0,4}, \quad a_{3,2} = 2k a_{1,3}, \quad a_{4,1} = k a_{2,2}, \\
&a_{5,0} = \frac{2k a_{3,1}}{5}
\end{aligned} \tag{12}$$

and our stream function looks like

$$\begin{aligned}
\psi = {} & a_{0,1} y - k a_{0,1} x^2 + a_{1,1} x y + a_{0,2} y^2 - 2/3\, k a_{1,1} x^3 - 2\, k a_{0,2} x^2 y \\
& + a_{1,2} x y^2 + a_{0,3} y^3 + k^2 a_{0,2} x^4 - 4/3\, k a_{1,2} x^3 y - 3\, k a_{0,3} x^2 y^2 \\
& + a_{1,3} x y^3 + a_{0,4} y^4 + \frac{8}{15}\, k^2 a_{1,2} x^5 + 3\, k^2 a_{0,3} x^4 y - 2\, k a_{1,3} x^3 y^2 \\
& - 4\, k a_{0,4} x^2 y^3 + a_{1,4} x y^4 + a_{0,5} y^5
\end{aligned} \tag{13}$$

(b) At the interface, surface tension contributes to the stress by, that is,

$$(\mathbf{n}.\sigma).\mathbf{t} = 0 \tag{14}$$

now the boundary condition can be expressed in terms of ψ as

$$-2\mu f^{'}(x)(\frac{\partial^2 \psi}{\partial x \partial y} + \mu \left(\frac{\partial^2 \psi}{\partial y^2} - \frac{\partial^2 \psi}{\partial x^2} \right) (1 - f^{'}(x)^2) = 0 \tag{15}$$

where $\sigma = -pI + [\nabla \mathbf{u} + (\nabla \mathbf{u})^T]$ is the stress tensor of the liquid, $\mathbf{n}$ is the normal vector and $\mathbf{t}$ is the tangent vector to the free surface. Insert the interface into (7) and (9) we have again some relations between the coefficients therefore the stream function reads

$$\begin{aligned}
\psi = {} & a_{0,1} y + k a_{0,1} x^2 + a_{1,1} x y + k a_{0,1} y^2 + \frac{2}{3} k a_{1,1} x^3 + 2\, k^2 a_{0,1} x^2 y \\
& + 4\, k a_{1,1} x y^2 + 2/3\, k^2 a_{0,1} y^3 + k^3 x^4 a_{0,1} + \frac{16}{3} k^2 a_{1,1} x^3 y \\
& + 2\, k^3 a_{0,1} x^2 y^2 + \frac{32}{3}\, k^2 a_{1,1} x y^3 + \frac{1}{3} k^3 a_{0,1} y^4 \\
& + \frac{32}{15}\, k^3 x^5 a_{1,1} + 2\, k^4 a_{0,1} x^4 y + \frac{64}{3}\, k^3 a_{1,1} x^3 y^2 \\
& + \frac{4}{3} k^4 a_{0,1} x^2 y^3 + \frac{64}{3}\, k^3 a_{1,1} x y^4 + \frac{2}{15} k^4 a_{0,1} y^5
\end{aligned} \tag{16}$$

and the last boundary condition is
(c) the normal stress condition

$$(\mathbf{n}.\sigma).\mathbf{n} = \frac{T}{R} \tag{17}$$

where $R = \frac{(1+f^{'}(x))^{\frac{3}{2}}}{f^{''}(x)}$ is the radius of curvature of the interface at $(x, f(x))$ and T is the surface tension, constant over the entire free surface. We can write the equation (11) as the following:

$$\begin{aligned}
&-p(1+f^{'}(x)^2)+2\mu f^{'}(x)^2\frac{\partial^2\psi}{\partial x\partial y}-2\mu\frac{\partial^2\psi}{\partial x\partial y}\\
&-2\mu f^{'}(x)\left(\frac{\partial^2\psi}{\partial y^2}-\frac{\partial^2\psi}{\partial x^2}\right)=\frac{f^{''}(x)T}{\sqrt{1+f^{'}(x)^2}}.
\end{aligned}\tag{18}$$

now the pressure coefficients must be determined. Let

$$p(x)=\sum_{i+j=0}^{\infty}p_{i,j}x^iy^j.\tag{19}$$

by applying the N.S.E. we can obtain the pressure series

$$\begin{aligned}
p=\;&c_{0,0}+\left(8\,k^2\mu\,a_{0,1}-\rho\,a_{0,1}a_{1,1}\right)x+\left(2\,\rho\,{a_{0,1}}^2k-12\,\mu\,ka_{1,1}\right)y\\
&+\left(48\,\mu\,k^2a_{1,1}-1/2\,\rho\,{a_{1,1}}^2\right)x^2+\left(16\,k^3\mu\,a_{0,1}-8\,k\rho\,a_{0,1}a_{1,1}\right)xy\\
&+\left(4\,k^2\rho\,{a_{0,1}}^2-48\,\mu\,k^2a_{1,1}-1/2\,\rho\,{a_{1,1}}^2\right)y^2\\
&+\left(\frac{32}{3}\,\mu\,k^4a_{0,1}-2/3\,\rho\,k^2a_{0,1}a_{1,1}\right)x^3+\left(384\,k^3\mu\,a_{1,1}-4\,k\rho\,{a_{1,1}}^2\right)x^2y\\
&+\left(16\,\mu\,k^4a_{0,1}-38\,\rho\,k^2a_{0,1}a_{1,1}\right)xy^2+\left(16/3\,k^3\rho\,{a_{0,1}}^2-128\,k^3\mu\,a_{1,1}-4\,k\rho\,{a_{1,1}}^2\right)y^3
\end{aligned}\tag{20}$$

if we insert this equation and (10) into (12) we can easily see that,

$$\begin{aligned}
a_{1,1}&=-\frac{p_{0,0}+2kT}{2\mu},\\
a_{2,1}&=\frac{1}{64}\,\frac{608\,k^3T\mu^2+4\,k^2\rho\,T^2+256\,k^2c_{0,0}\mu^2+192\,a_{3,0}\mu^3k+4\,\rho\,Tc_{0,0}k+\rho\,{c_{0,0}}^2}{\mu^3k}
\end{aligned}\tag{21}$$

As you can see these two coefficients depend on the first pressure, k, T and μ. The interface will be transform to the straight line after a coordinate change $x=\xi$ and $y=\eta+f(x)$ and substitute into stream function we get

$$\begin{aligned}
\psi=\;&a_{0,1}\eta+ka_{0,1}\eta^2+2\,ka_{0,1}\xi^2+a_{1,1}\xi\,\eta+\frac{5}{3}ka_{1,1}\xi^3\\
&+a_{2,1}\eta\,\xi^2+4\,ka_{1,1}\xi\,\eta^2+a_{0,3}\eta^3\\
&+4\,k^3\xi^4a_{0,1}+\frac{40}{3}\,k^2a_{1,1}\xi^3\eta+4\,k^3a_{0,1}\eta^2\xi^2\\
&+\frac{32}{3}\,k^2a_{1,1}\xi\,\eta^3+\frac{1}{3}k^3a_{0,1}\eta^4\\
&+\frac{172}{15}\,k^3\xi^5a_{1,1}+8\,k^4a_{0,1}\eta\,\xi^4\\
&+\frac{160}{3}\,k^3a_{1,1}\xi^3\eta^2+\frac{2}{15}k^4a_{0,1}\eta^5\\
&+\frac{8}{3}k^4a_{0,1}\eta^3\xi^2+\frac{64}{3}\,k^3a_{1,1}\xi\,\eta^4
\end{aligned}\tag{22}$$

if we write the velocities,

$$
\begin{aligned}
\dot{\xi} &= a_{0,1} + 2\,k a_{0,1}\eta + a_{1,1}\xi + 4\,k^2 a_{0,1}\xi^2 + 8\,k a_{1,1}\xi\,\eta + 2\,k^2 a_{0,1}\eta^2 \\
&\quad + \frac{40}{3}\,k^2 a_{1,1}\xi^3 + 8\,k^3 a_{0,1}\eta\,\xi^2 + 32\,k^2 a_{1,1}\xi\,\eta^2 \\
&\quad + 4/3\,k^3 a_{0,1}\eta^3 + 8\,k^4 a_{0,1}\xi^4 + \frac{320}{3}\,k^3 a_{1,1}\xi^3\eta \\
&\quad + 2/3\,k^4 a_{0,1}\eta^4 + 8\,k^4 a_{0,1}\eta^2\xi^2 + \frac{256}{3}\,k^3 a_{1,1}\xi\,\eta^3 \\
\dot{\eta} &= -4\,k a_{0,1}\xi - a_{1,1}\eta - 5\,k a_{1,1}\xi^2 - 8\,k^2 a_{0,1}\eta\,\xi - 4\,k a_{1,1}\eta^2 \\
&\quad - 16\,k^3\xi^3 a_{0,1} - 40\,k^2 a_{1,1}\xi^2\eta - 8\,k^3 a_{0,1}\eta^2\xi - \frac{32}{3}\,k^2 a_{1,1}\eta^3 \\
&\quad - \frac{172}{3}\,k^3\xi^4 a_{1,1} - 32\,k^4 a_{0,1}\eta\,\xi^3 - 160\,k^3 a_{1,1}\xi^2\eta^2 \\
&\quad - 16/3\,k^4 a_{0,1}\eta^3\xi - \frac{64}{3}\,k^3 a_{1,1}\eta^4
\end{aligned}
\tag{23}
$$

The linear approximation of the equations for the streamlines is

$$
\begin{pmatrix} \dot{\xi} \\ \dot{\eta} \end{pmatrix} = \begin{pmatrix} a_{0,1} \\ 0 \end{pmatrix} + \begin{pmatrix} a_{1,1} & 2ka_{0,1} \\ -4ka_{0,1} & -a_{1,1} \end{pmatrix} \begin{pmatrix} \xi \\ \eta \end{pmatrix} \tag{24}
$$

with the Jacobian

$$
J = \begin{pmatrix} a_{1,1} & 2ka_{0,1} \\ -4ka_{0,1} & -a_{1,1} \end{pmatrix} \tag{25}
$$

If $a_{0,1} = 0$, the origin is critical point. Substitute $a_{0,1} = 0$ into Jacobian we can say the origin is a saddle with eigenvalues $a_{1,1}$ and $-a_{1,1}$. The separatrix corresponding to $a_{1,1}$ is the interface, while the other seperatrix is dividing streamline which is orthogonal the interface given by $\xi = 0$.

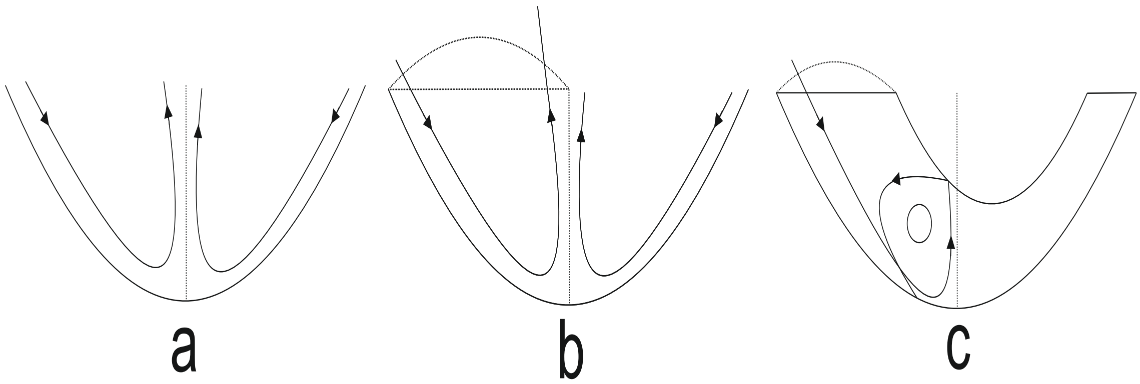

Figure 16: Non-degenerate critical point

In figure $(2)-a$, shows the moving of the velocity to us when $a_{1,1} \neq 0$. In figure $(2)-b$ we consider the inlet diameter is $\frac{x_0}{2}$, then the flow turn back. So the first experiments to us, the inlet diameter must be smaller than $\frac{x_0}{2}$ when $a_{1,1} \neq 0$. In figure $(2) - c$

we consider the inlet diameter is smaller than $\frac{x_0}{2}$, then we need to find the velocity behaviour under the upper-wall. And we know that if the flow hit the upper-wall, the velocity of the flow will decrease and the performance of the machine also will decrease. So our question is which condition is the best for us? Firstly we will see that when $a_{1,1} \neq 0$, it is not very useful for us, so we need to take $a_{1,1} = 0$ then we can look at the new non-degeneracy condition. If $a_{0,1} = a1, 1 = 0$ which means physically that $p_{0,0} = 2kT$ from the normal stress condition, the origin is a degenerate critical point, since zero is a double eigenvalue of J. The corresponding streamfunction are given by

$$\psi = a_{2,1}\xi^2\eta + a_{0,3}\eta^3 + O(\xi^4, \eta^4) \tag{26}$$

For $a_{2,1}/a_{0,3} < 0$ there are two seperatrix and $a_{2,1}/a_{0,3} > 0$ we have no seperatrix they are shown in the figure (3), respectively.

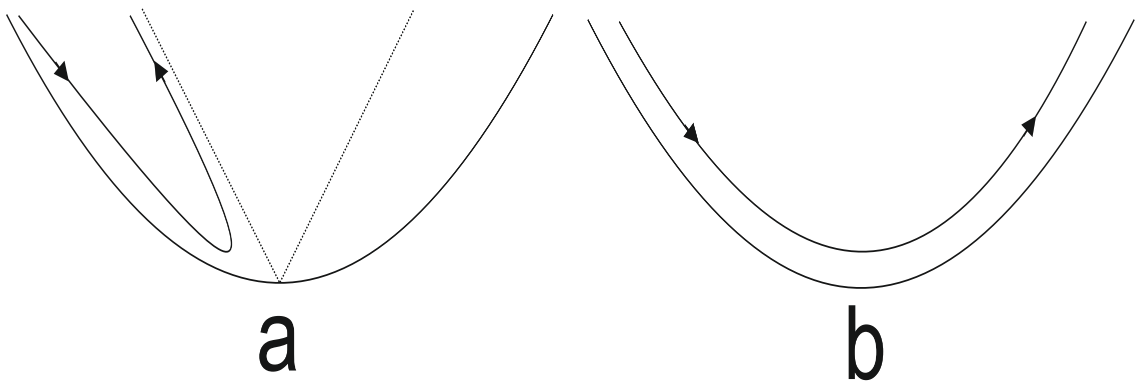

Figure 17: simple degenerate critical point

As we can see, the figure (3) implies to us when $a_{2,1}/a_{0,3} > 0$ flow motion is useful. But we are looking for the best way and with using the flow velocity we can create a pressure and we can create the narrow region. The question is can we success or not?

Yes. Remember that we use the simple degeneracy condition. If we unfold the double degeneracy condition we can get best result.

B Matlab code for the 3D prototype

```
%constants
N = 200;  %resolution (the bigger the better)
S = 1.6;  %the biggest value of parameter s
A = 3;    %the biggest value of parameter a
t = .1;   %thickness
```

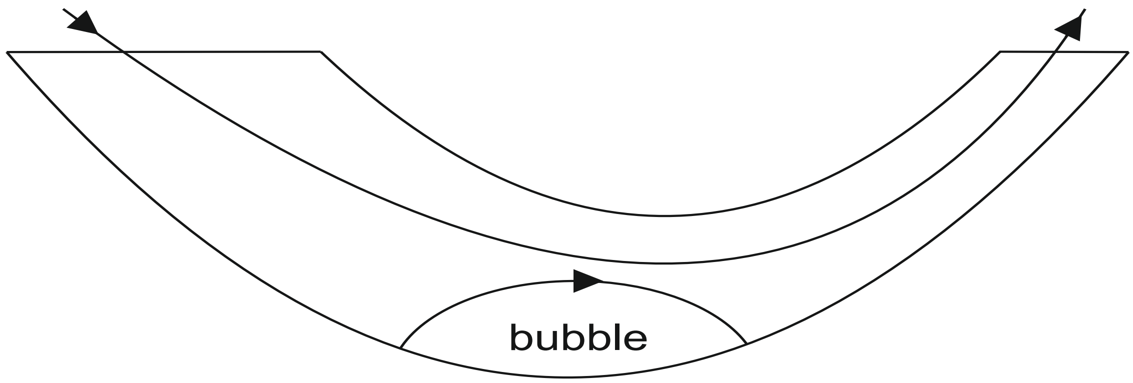

Figure 18: The interface with bubble

```
d = 5;     %scale (to stl format)
l = 5;     %distance between centers of input hole and output hole)

%parameters
alpha           = (linspace(0,2*pi,N));         %first torus angle
beta_            = (linspace(0,pi,N));          %second torus angle
s               = repmat(linspace(0,S,N),N,1); %to radius function
a               = repmat(linspace(0,A,N),N,1); %to curve function
[alpha, beta_] = meshgrid(alpha,beta_);        %alpha, beta -> matrices

%function for radius of cross-section
r  = (1 - 1/2 .* s)'; %inner pipe
r1 = r + t;           %outer pipe

%function for curve for both surfaces
R = (4 + 1/2 .* a)';

%coordinates of inner surface based on torus parametrization
x = (R + r .* cos(alpha)) .* cos(beta_);
y = (R + r .* cos(alpha)) .* sin(beta_);
z = r .* sin(alpha);

%coordinates of outer surface based on torus parametrization
x1 = (R + r1 .* cos(alpha)) .* cos(beta_);
y1 = (R + r1 .* cos(alpha)) .* sin(beta_);
z1 = r1 .* sin(alpha);

%whole model
X = [x; flipud(x1); x(1,:)] .* l ./ 10;
Y = [y; flipud(y1); y(1,:)] .* l ./ 10;
Z = [z; flipud(z1); z(1,:)] .* l ./ 10;

%plotting
```

```
figure; grid on;
mesh(X,Y,Z); axis equal;

%scaling
X = X ./ d;
Y = Y ./ d;
Z = Z ./ d;

%converse to stl
surf2stl('design.stl',X,Y,Z);
```

References

[1] J. Cimbala and Y. Cengel, *Essentials of Fluid Mechanics*, Fundamentals and Applications, McGraw Hill Higher Education, 2008.

[2] S. S. Cook, *Erosion by Water-Hammer*, Proceedings of the Royal Society of London. Series A, Containing Papers of a Mathematical and Physical Character, 119 (1928), pp. 481–488.

[3] J. Eggers and E. Villermaux, *Physics of liquid jets*, Reports on Progress in Physics, 71 (2008), p. 036601.

[4] V. Ferreira, M. Tomé, N. Mangiavacchi, A. Castelo, J. Cuminato, A. Fortuna, and S. McKee, *High-order upwinding and the hydraulic jump*, International journal for numerical methods in fluids, 39 (2002), pp. 549–583.

[5] J. Hearfield, *Water Flowing in Pipes*. http://www.johnhearfield.com/Water/Water_in_pipes2.htm#H4. Accessed: August 2013.

[6] ITACA, *Fluid Mechanics For Gravity Flow Water Systems and Pumps*. http://www.itacanet.org/fluid-mechanics-for-gravity-flow-water-systems-and-pumps/appendix-3-friction-losses-and-the-reynolds-number/. Accessed: August 2013.

[7] X. Liu, J. Lienhard, and J. Lombara, *Convectiwe heat transfer by impingement of circular liquid*, Journal of heat transfer, 113 (1991), pp. 571–582.

[8] É. Lorenceau, D. Quéré, and J. Eggers, *Air Entrainment by a Viscous Jet Plunging into a Bath*, Phys. Rev. Lett., 93 (2004), p. 254501.

[9] B. J. McKeon, C. J. Swanson, M. V. Zagarola, R. J. Donnelly, and A. J. Smits, *Friction factors for smooth pipe flow*, J. Fluid Mech., 511 (2004), pp. 41–44.

[10] G. Mohiuddin Mala and D. Li, *Flow characteristics of water in microtubes*, International Journal of Heat and Fluid Flow, 20 (1999), pp. 142–148.

[11] L. F. Moody, *Friction factors for pipe flow*, Trans Asme, (1944).

[12] T. Obara, N. K. Bourne, and J. E. Field, *Liquid-jet impact on liquid and solid surfaces*, Wear, 186 (1995), pp. 388–394.

[13] M. Sibulkin, *Transition from Turbulent to Laminar Pipe Flow*, Physics of Fluids, 5 (1962), p. 280.

[14] V. K. Vasista, *Experimental study of the hydrodynamics of an impinging liquid jet*, PhD thesis, Massachusetts Institute of Technology, 1989.

[15] K. Watanabe, Y. Udagawa, and H. Udagawa, *Drag reduction of newtonian fluid in a circular pipe with a highly water-repellent wall*, Journal of Fluid Mechanics, 381 (1999), pp. 225–238.

[16] Water Efficiency Labelling and Standards (WELS) scheme, *Water efficiency*. http://www.waterrating.gov.au/consumers/water-efficiency. Accessed: August 2013.

[17] WaterSense, U.S. Environmental Protection Agency, *WaterSense Labeled High-Efficiency Lavatory (Bathroom Sink) Faucet Specification*. http://www.epa.gov/WaterSense/pubs/faq_bs.html. Accessed: August 2013.

[18] F. M. White, *Fluid Mechanics*, McGraw Hill Higher Education, 6th ed., 2008.