Chapter 1

The experiment

Segregation effect on a granular free-surface flow going down a slope.

Segregation effects in granular avalanches are important as they influence overall flow characteristics. Whilst other factors such as density do play a role, the effect of size-segregation is often most striking and is what we will investigate here. The formation of lobate structures bounded by coarse-particle-rich lateral levees is significant: channelising the flow, such structures enhance overall run-out distance, meaning that in natural avalanches, the potential danger is increased. Those features have been observed in the field as well as in laboratory experiments. Figure 1.1 shows two such examples.

Whilst the remains shown in figure 1.1 provide valuable insight into flow behaviour and run-out characteristics, one can still only speculate about the processes involved before the material came to rest. Observing natural granular flows as they occur is impractical, due to unpredictability and potential dangers. These large scale features can be qualitatively reproduced in the laboratory. However the effect of pore pressure in natural debris flows is under-represented by geometrically similar small-scale experiments, while the effects of inertia and viscosity are over-represented. These phenomena may have a strong impact on segregation and deposition effects. Therefore it is best to simulate these debris flows by large-scale experiments.

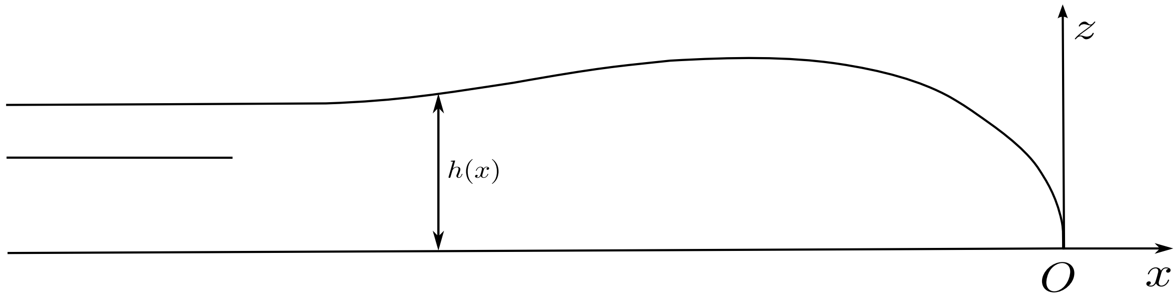
1.1 The experiment

It is what was done in a series of two experiments in August 2011 at the United States Geophysical Survey (USGS) debris-flow flume, near Blue River, Oregon [?].

The experimental device consists of a 95 meters long, 2 meters wide inclined channel terminated by a 25 meters long roughly horizontal run-out pad. The mixture of water saturated sand (0.0625-2 mm) and gravel (2-32 mm) is prepared behind the gates closing the channel entrance. When the 10 m^3 mixture is released, it accelerates and reach the end of the channel after $\approx$ 10 seconds. Just after the head of the flow pass the end of the channel, it is truncated and the rest of the flow is diverted. A measurement of the head's surface velocity is performed by ultra-rapid cameras for the first few meters. During the measurement, the head has a constant velocity of 2.0 $m \cdot s^{-1}$. It then slows down; after a few more seconds the material is deposited on the run-out pad. The rest of the flux is diverted to prevent it to mix with the head on the run-out pad and bury the initial deposit.

We are interested in studying the last stage of the avalanche, during which the cut head front is going at constant speed on the run-out pad. In the reference frame travelling at the front speed, we observe that the velocity field is stationary. So in that frame, particle paths coincides with streamlines.

The surface of the flow is composed of large gravel particles. This is an evidence that segregation effects are at play. A key feature of the experimental flow, also observed in natural debris flow, is the existence of high vertical shearing rate (fig 1.1). The uppermost layer of particles is going faster than the internal layers. Therefore large particles constituting it will arrive at the front of the flow head and will be buried. Because of segregation effects, they will be push up. Because of the existence of a high lateral shearing rate, as they go up they will also be pushed to the sides and deposit inside lateral


z
h(x)
O
x

levees. This is this recirculation we are interested in studying.

1.2 Modelling the experiment

An option would be to model the experimental flow using fluid equations. Prescribing the initial velocity and mass distribution would enable us to deduce the whole evolution of the velocity field. However, it is not trivial to choose a rheology (ie a granular friction law) adapted to the problem, and in the same time not leading to ill-posedness. And since measurement of surface velocity and internal shear stress were performed during the experiments, it is natural to prescribe a velocity field agreeing with the measurements, and use it to analyse what is going on inside the flow. As we said before, during the phase we analyse, the velocity field is stationary in the travelling frame. So we will do all of our analysis in this frame.

1.2.1 Constructing the velocity field

To begin with, we prescribe the shape of the flow surface (1.2.1). There is a general procedure to build a velocity field knowing the flow surface and base, and the *velocity profile*. This quantity is defined as

$$h(x,y)\bar{u}(x,y) = \int_{z=0}^{z=h} u(x,y,z)dz \tag{1.1}$$

Rather than manipulating the velocity components $u(x,y,z)$, $v(x,y,z)$, $w(x,y,z)$, we will make use of the *depth integrated velocity components* $\bar{u}(x,y)$, $\bar{v}(x,y)$, $\bar{w}(x,y)$. They are defined by

$$h(x,y)\bar{u_i}(x,y) = \int_{z=0}^{z=h} u_i(x,y,z)dz \tag{1.2}$$

Since they only depend on 2 variables instead of 3, they are easier to deal with. In particular, since we assume 3D incompressibility, we have

$$\frac{\partial}{\partial x}h\bar{u} + \frac{\partial}{\partial y}h\bar{v} = 0 \tag{1.3}$$

We can thus define $\bar{u}$ and $\bar{v}$ by a single scalar field: the stream function $\psi(x,y)$, such that

$$\frac{\partial \psi}{\partial x} = -h\bar{v} \tag{1.4}$$

$$\frac{\partial \psi}{\partial y} = h\bar{u} \tag{1.5}$$

A stream function producing a velocity field in good agreement with the experiments is

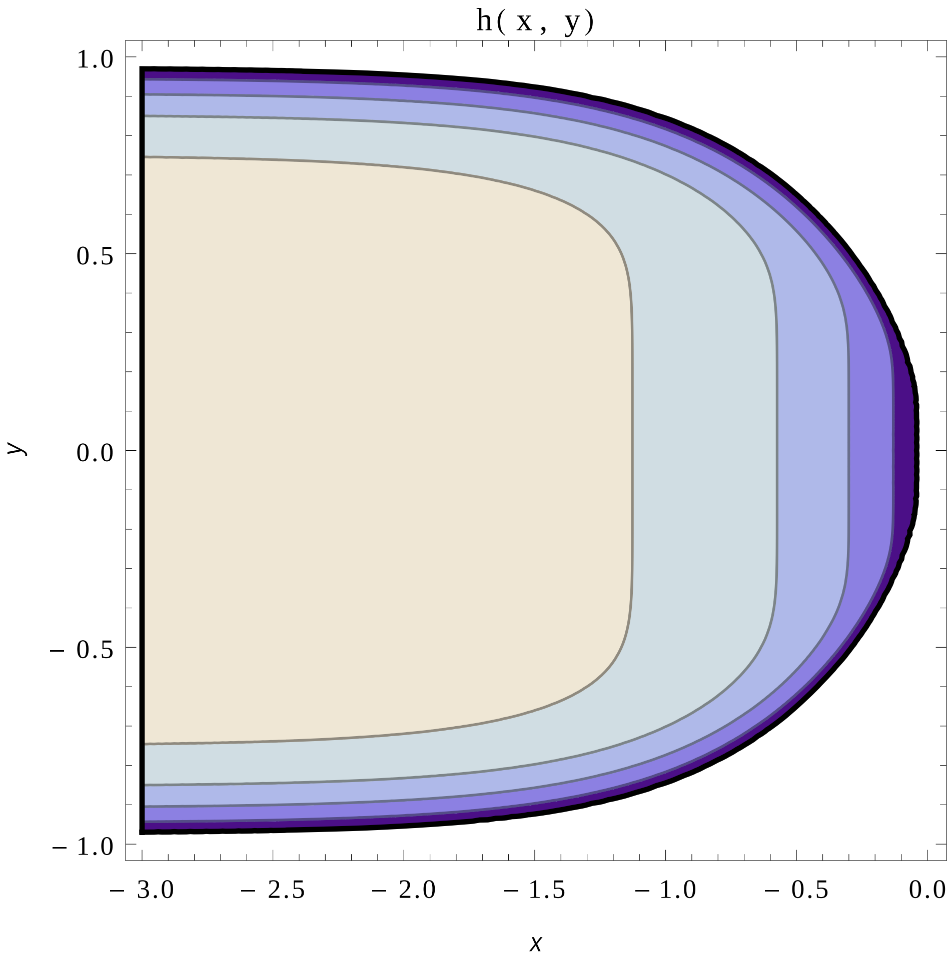
$$\psi(x,y) = \frac{HU}{W^2}\left(kyy_0^2 - \frac{k}{2n+1}\frac{y^{2n+1}}{y_0^{2n-2}} - \frac{1}{2m+1}\frac{y^{2m+1}}{y_0^{2m-2}} + \frac{1}{2n+2m+1}\frac{y^{2n+2m+1}}{y_0^{2n+2m-2}}\right) \tag{1.6}$$

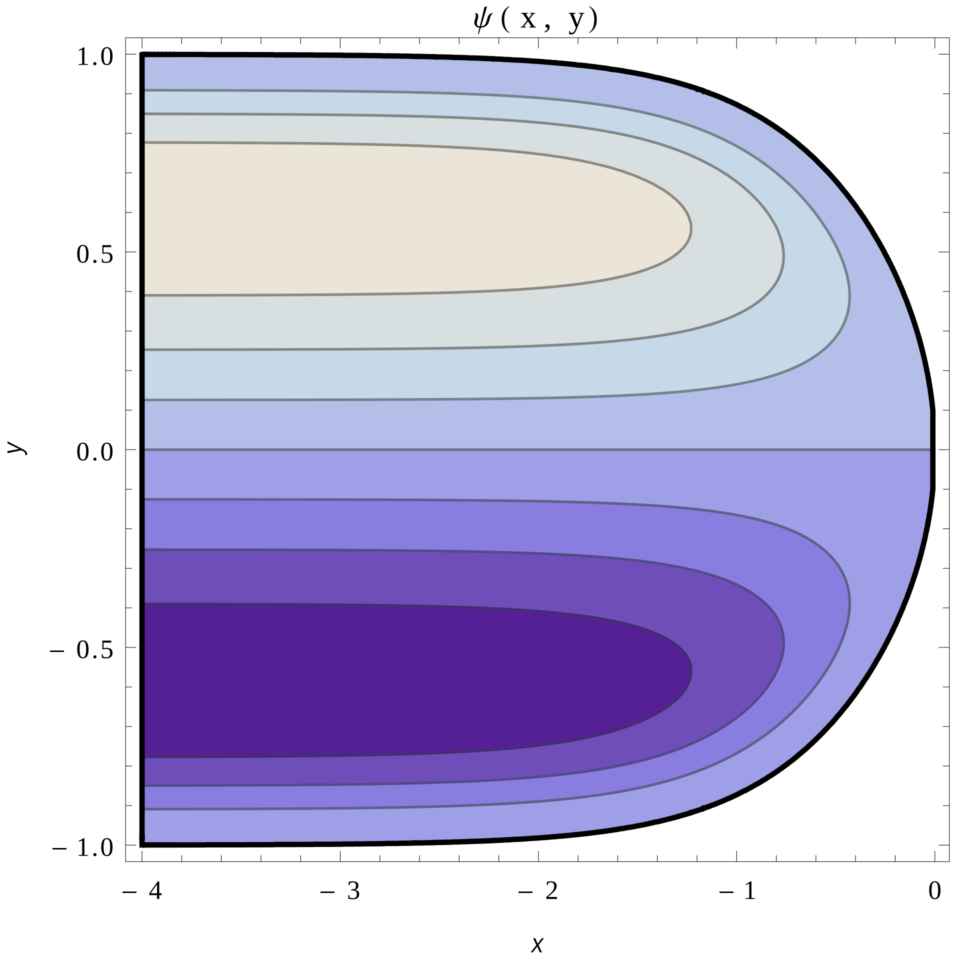
where H is the height of the flow far from the head, and W and U respectively the typical width and speed of the flow. They are chosen to fit the experimental data.

We have prescribed the depth-integrated velocity field. To compute the full velocity field, we still need a supplementary information, which is provided by the *velocity profile*, containing the z dependency of the velocity components It is defined by[1]

$$\begin{pmatrix} u(x,y,z) \\ v(x,y,z) \end{pmatrix} = f\left(\frac{z}{h}\right)\begin{pmatrix} \bar{u}(x,y) \\ \bar{v}(x,y) \end{pmatrix} \tag{1.7}$$

[1]Note that this definition of f implies that u and v have the same z dependency. Though it is not necessarily the case, it has proven a good approximation for the flow we seek to model.


h(x , y)
1.0
0.5
0.0
− 0.5
− 1.0
y
− 3.0
− 2.5
− 2.0
− 1.5
− 1.0
− 0.5
0.0
x


ψ (x, y)
1.0
0.5
0.0
− 0.5
− 1.0
y
− 4
− 3
− 2
− 1
0
x

Now we have the full velocity profile. We can use it to deduce the evolution of the concentration in small and large particle using the segregation equation we derived in chapter 1:

$$\frac{\partial}{\partial t}\phi + \frac{\partial}{\partial x}u\phi + \frac{\partial}{\partial y}v\phi + \frac{\partial}{\partial z}w\phi - q\frac{\partial}{\partial z}\phi(1-\phi) = 0 \tag{1.8}$$

Solving this equation analytically is challenging. However, since it is a conservation law, we do have a number of tools we can use to deal with the problem. We will talk about this in the next chapter. For a start, we would like to solve this equation numerically, to have an idea of its structure.

1.2.2 Numerical simulations

Solving conservation laws numerically is a challenging problem. These equations describe quantities that can become discontinuous. For example, a fully segregated granular material is composed of a layer of large particles on top of a layer of small particles. Thus $\phi = 0$ in the upper part of the system, and $\phi = 1$ in the lower part. The two domains are separated by a concentration jump.

Such jumps are difficult to render numerically, as numerical viscosity tends to smooth out discontinuities.

Various approach can be used to solve numerically conservation laws. We choose to handle the problem using a *finite volume method*. Details about the particular finite volume method we use can be found in APPENDIX ??. Finite volume methods are exact in the sense that they conserve the solved quantity. TODO: graphic showing how the numerical fluxes cancel each other out except at the boundaries of the domain. For us, that means that the *numerically computed* total concentration in small particles in the integration domain coincides with the *exact* total concentration, at any time. This property of conservation is extremely important: it ensures that discontinuities will be accurately calculated by our scheme.[2]

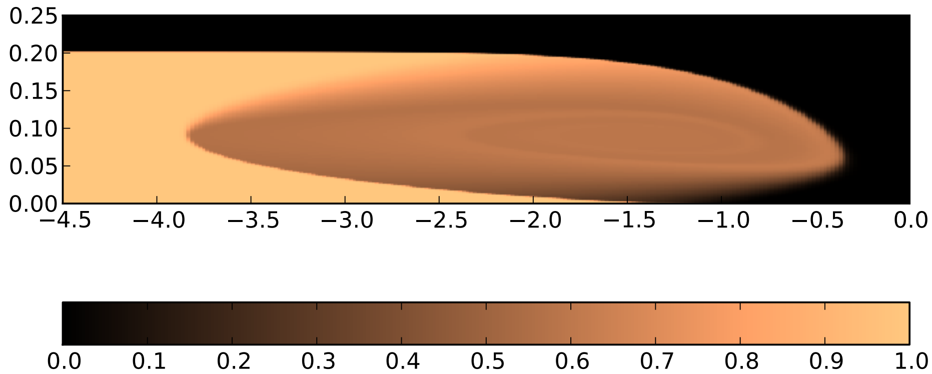
Far from the head the mixture is fully segregated: it is composed of a layer of large particles (20% of the total flow height) in top of a layer of small particles (80% of the total height). They are separated by a sharp discontinuity in the concentration ϕ, called a *shock wave*. Close to the head, because of the shearing process, the layer of large particles wraps around the layer of smalls. This gives rise to a complex structure, which we would like to explore numerically. We set the initial particles to occupy the 0.8 lowest fraction of the flow height. We then let the system evolve until it reach a steady state. Before we simulate the whole $3D$ flow, we will start with a 2D simulation in the centre plane, ie the plan $y = 0$. Indeed in this plane, $\partial v/\partial y = 0$, which means that particles in the centre plane stay in the centre plane. We can rewrite the segregation equation:

$$\frac{\partial}{\partial t}\phi + \frac{\partial}{\partial x}u\phi + \frac{\partial}{\partial z}w\phi - q\frac{\partial}{\partial z}\phi(1-\phi) = -v\frac{\partial}{\partial y}\phi \tag{1.9}$$

The term $-v\partial\phi/\partial y$ can be seen as a source term in the 2D problem. The resulting structure is shown in fig 1.2.2.

We observe a spiralling structure, which we will try to explain and reproduce analytically in the next chapter.

[2] It can be easily shown (Lax-Wendroff theorem CIT) that finite-volume schemes, provided that they converge, will converge to a discontinuous solution of the *exact* conservation law, while non conservative schemes whilst doing as well as finite-volume methods every time the solution is continuous, will cease to be accurate if the solution has discontinuities. Since discontinuities are an essential feature of conservation laws, the conservative property of finite-volume methods is of the highest importance.


0.25
0.20
0.15
0.10
0.05
0.00
−4.5
−4.0
−3.5
−3.0
−2.5
−2.0
−1.5
−1.0
−0.5
0.0
0.0
0.1
0.2
0.3
0.4
0.5
0.6
0.7
0.8
0.9
1.0