CFD : falling film flow

For the TER M1

Abstract

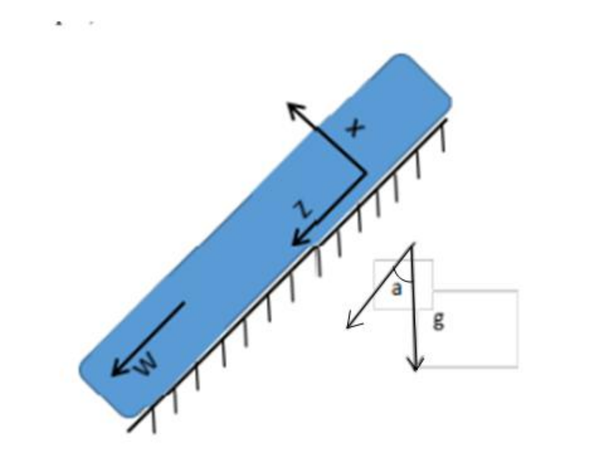
Rain water is often observed to flow down nearly as a sheet down an inclined surface, for instance,on cars in heavy rain. In industry such flows also occur, notably for mass transport applications and coating processes. The flow regime may be desirable or not, depending on the application, and this is a design consideration in industry.

It is proposed here to conduct CFD simulations for a simplified model system of a liquid film moving down an inclined simple substrate. Two-dimensional flow will be assumed and some further reductions will be made, such that the simulations can be run on a PC, with software such as Fluent. The corresponding system in three dimensions is shown in the figure.

The objective here is to determine how the flow and liquid layer shape change with flow parameters. Also, we would like to determine whether it is possible to use the CFD software to simulate such flows. It will be possible to compare against prior results.

This project may be of interest to students in mechanics who would like to familiarize themselves better with CFD, and with the main concepts in two-phase flows.

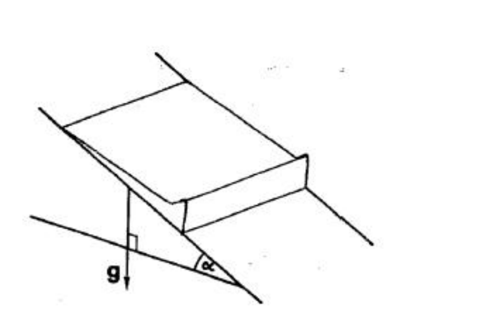
Introduction



Analytical Solution

To solve this problem , we have to determiner the vitess w(x) , we start with the z-compoment of momentum equation :

$$\frac{\partial w}{\partial t} + \mu \frac{\partial w}{\partial x} + w \frac{\partial w}{\partial z} = -\frac{1}{\rho}\frac{\partial P}{\partial z} + \nu\left(\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial z^2}\right) + g_z$$



Assumption :

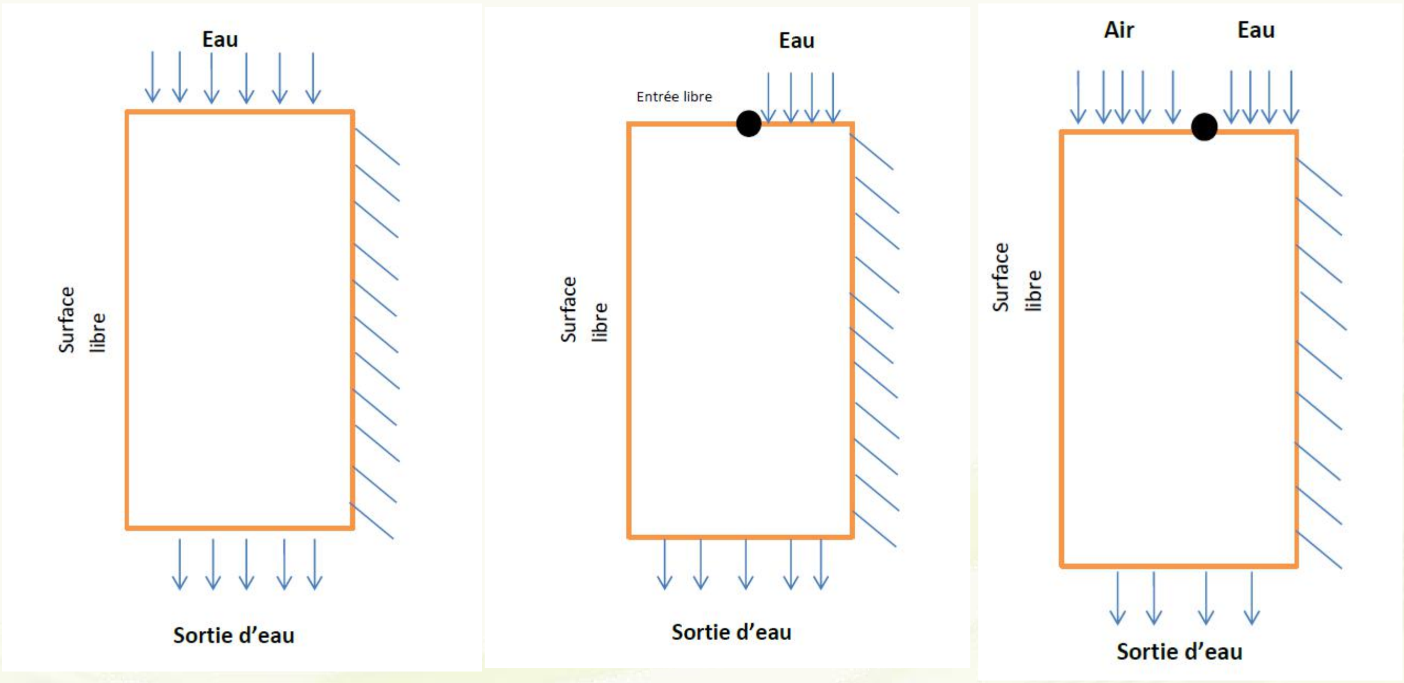
$$0 = -\frac{1}{\rho}\frac{\partial P}{\partial z} + \nu \frac{\partial^2 w}{\partial x^2} + g \cos \alpha$$

h is the thickness of the fluid, we have :

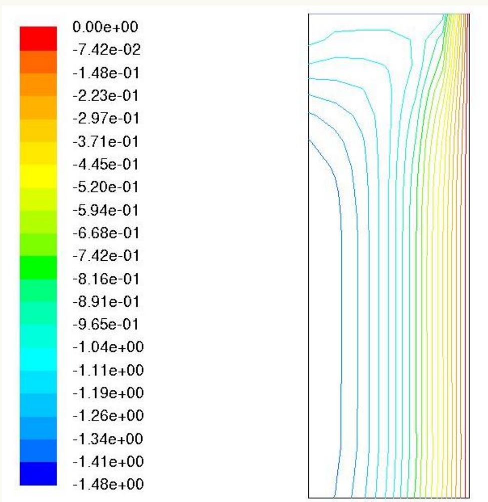
$$w(x)=\frac{g\cos\alpha}{2}\left(hx-\frac{x^2}{2}\right)$$

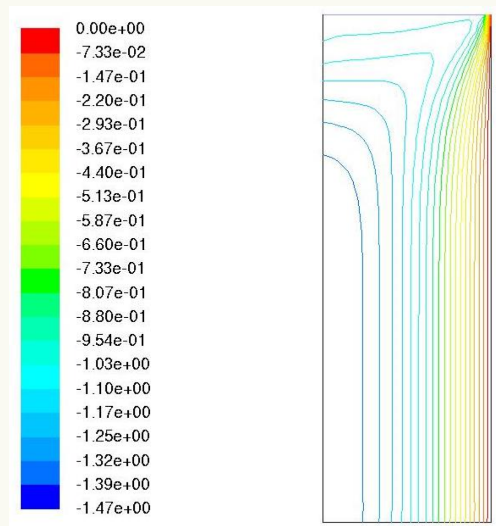
For the numerical analysis, we will use the Fluent software. To simplify our research first, the study will break down into three major parts. And we do the three simulations in the case of the non-inclined plane (that is, we do not decompose gravity). We will do the simulation in reality situation in the following.

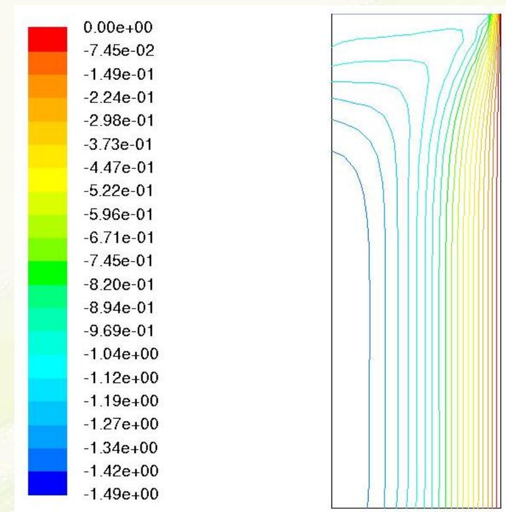
Fluent


Eau
Surface libre
Sortie d'eau
Eau
Entrée libre
Surface libre
Sortie d'eau
Air
Eau
Surface libre
Sortie d'eau

We use three meshes to make this first simulation: a fine mesh, a medium, and a coarse. The three are with the same density, the same density, and the same input speed initialized to 1.

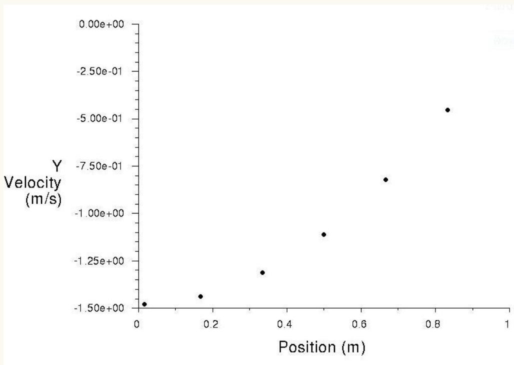


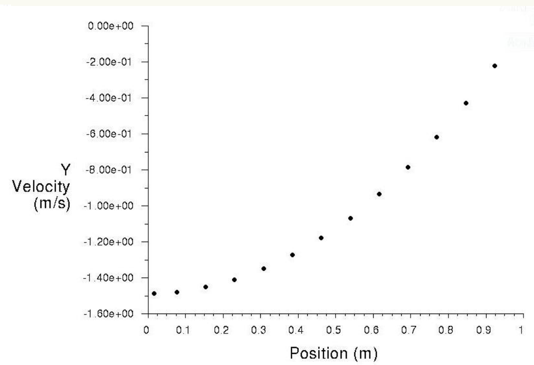


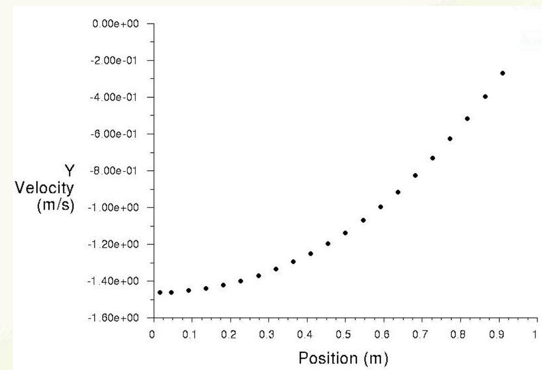


fin midium gross

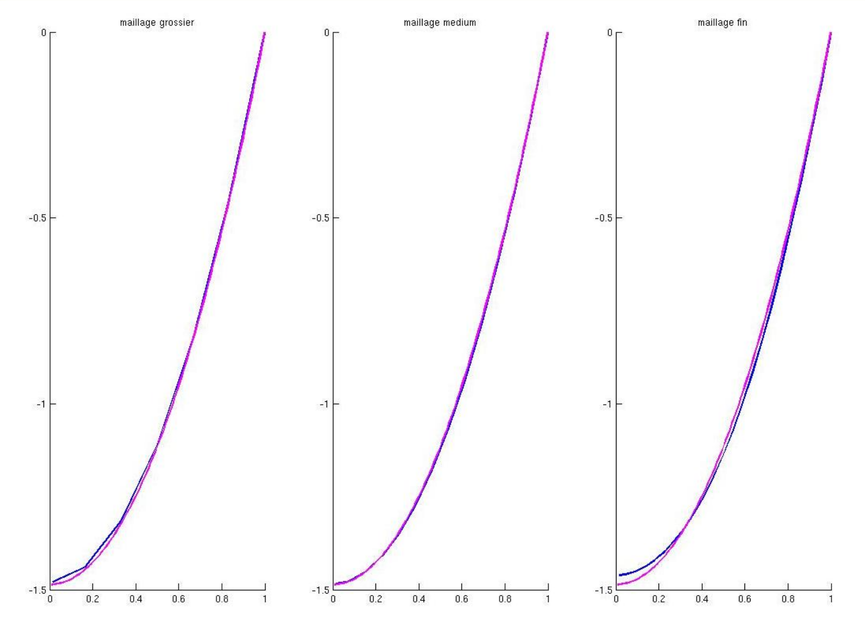
The numerically obtained velocity profile is a parabolic profile, which corresponds to a conventional flow in a pipe, with a zero velocity at the wall (position = 1), and a certain velocity of about 1.5 m / s to the left of The flow (explains the reason)





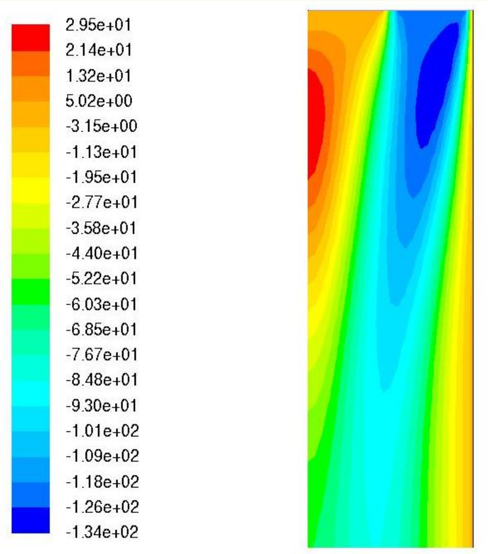


Error

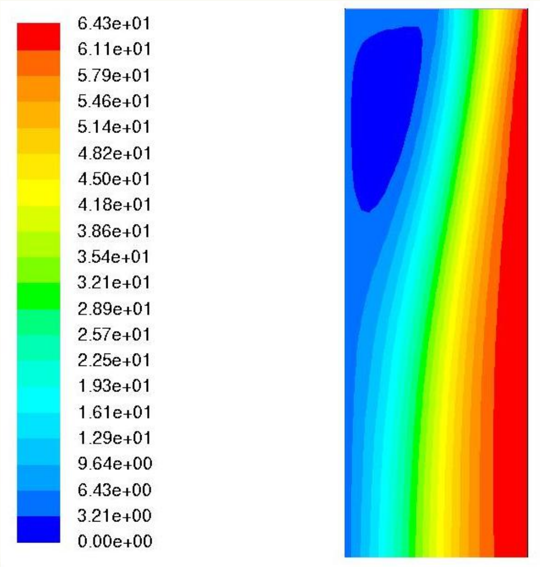


We notice a good superposition of the two solutions for the medium mesh. On the other hand, for the fine mesh, there are slight differences between the two curves, notably near the interface where the difference becomes remarkable and not negligible. The coarse mesh is correct to the right of the flow, but begins to diverge from the exact solution when the position tends to 0.

Re=40

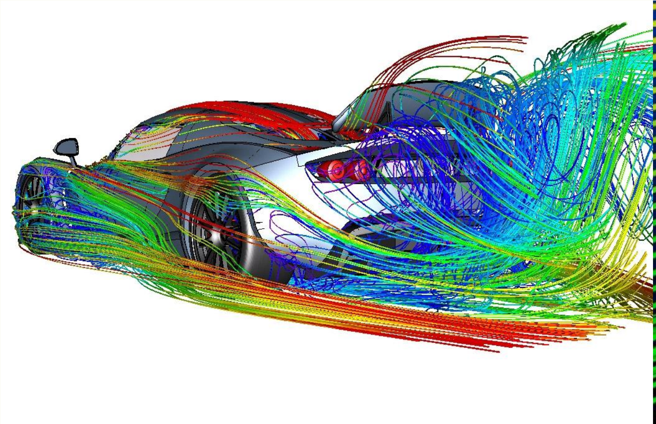


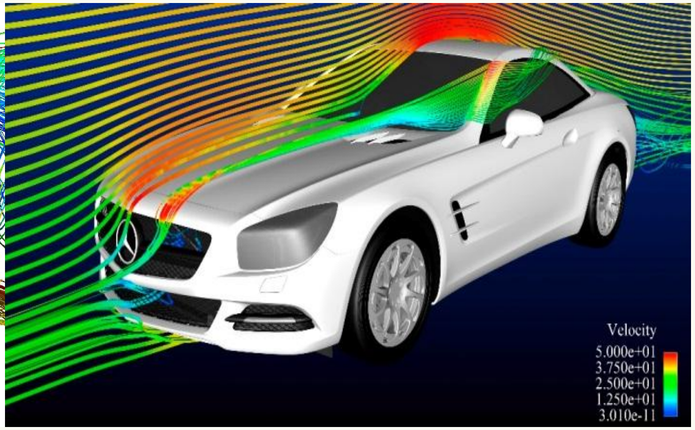
y-vitess profile



stream function

Application





有个peter举得
例子，我忘了
啥了，什么jet?

hhhh

hhh

hhhh

Conclusion

hhhhh

Thanks !