A 1D numerical model for incompressible fluids in pipes with variable section

Elena Travaglia

Ph.D. Modeling and Data Science
Università degli studi di Torino

In collaboration with Ferrero s.p.a of Alba

Tutor: Matteo Semplice
Università degli studi dell'Insubria

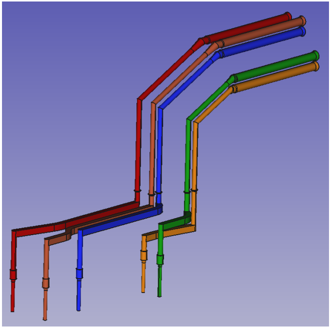
11-02-2021

The industrial setting

We have an incompressible fluid and we want to compute the pressure drops in channels in a very efficient way.

Our task is to find a model

- with a very low computational cost.
- that represents all the fluid and the geometry characteristics.
- at least of third order of accuracy.



Mathematical model

The mass and the momentum conservation equations are

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \bar{u}) = 0 \qquad (1)$$

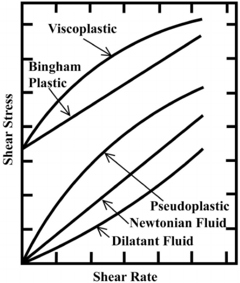
$$\rho \left(\frac{\partial \bar{u}}{\partial t} + (\bar{u} \cdot \nabla) \bar{u} \right) - \rho g = -\nabla p + \nabla \cdot \sigma \qquad (2)$$

Assumptions:

- incompressible fluid $\rho = cost$ so (1) begins $\nabla \cdot \bar{u} = 0$
- "no-slip" condition so $\bar{u} = 0$ on solid boundary
- inlet : Dirichlet condition for $\bar{u}$
  outlet: Neumann condition for p.

A fluid can be

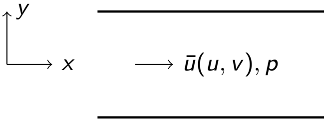
- Newtonian $\quad \sigma = \mu \bar{\gamma}$
- Non-Newtonian $\quad \sigma = \mu(\bar{\gamma})\,\bar{\gamma}$
  where $\bar{\gamma}$ is the tensor strain and it is defined as $\bar{\gamma} = \nabla \bar{u} + (\nabla \bar{u})^T$



NS analytic solution, Poiseuille

The incompressible Navier-Stokes equations:

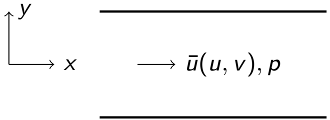
$$\begin{cases} \rho\left(\dfrac{\partial \bar{u}}{\partial t} + (\bar{u}\cdot\nabla)\bar{u}\right) = -\nabla p + \nabla\cdot\sigma \\ \nabla\cdot\bar{u} = 0 \end{cases}$$



NS analytic solution, Poiseuille

The incompressible Navier-Stokes equations:

$$\begin{cases} \rho\left(\dfrac{\partial \bar{u}}{\partial t} + (\bar{u}\cdot\nabla)\bar{u}\right) = -\nabla p + \nabla\cdot\sigma \\ \nabla\cdot\bar{u} = 0 \end{cases}$$



Hypothesis:

- the radius R is constant
- stationary solution
- fully developed flow

$\Longrightarrow$

- $v = 0$
- $u(x) = \text{const}$

$$\Longrightarrow \quad \partial_y\left(\mu\,\partial_y u\right) = \partial_x p$$

elliptic eq. on the transverse direction

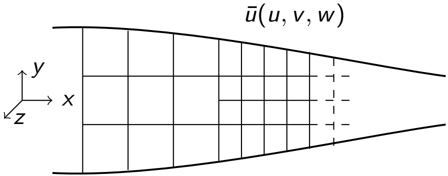
$\Downarrow$

we obtain a velocity profile, that it is parabolic for a newtonian fluid

$$u(y) = \frac{\partial_x p}{4\mu}\left(R^2 - y^2\right)$$

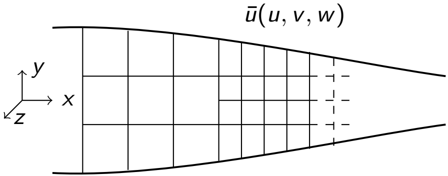
If R is non constant

- $v, w \neq 0$
  $\Rightarrow \partial_y p \neq \partial_z p \neq 0$
- $u(x) \neq$ constant
- $u(y) \neq$ parabolic
- the pressure drop isn't constant
  $\Rightarrow \partial_x p \neq constant$



If R is non constant

- $v, w \neq 0$
  $\Rightarrow \partial_y p \neq \partial_z p \neq 0$
- $u(x) \neq$ constant
- $u(y) \neq$ parabolic
- the pressure drop isn't constant
  $\Rightarrow \partial_x p \neq constant$



We could :

- use a 3D solver
- discretize in x and in the transversal directions
- approximate the solution with $\mathbb{P}_g$ in each cell

Problem: it requires a high computational cost.

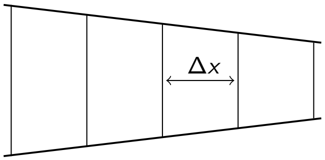
Almost 1D approximation

Idea: we want to solve a 1D equation in the x direction, but we don't know how to analytically calculate the velocity profile, so we decide to compute it numerically.

We assume:

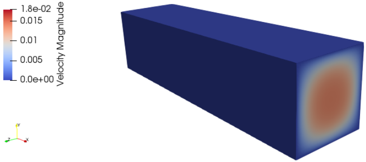
- slowly varying diameter of pipe
- $v, w = 0 \quad \Longrightarrow \quad \frac{\partial p}{\partial_y} = \frac{\partial p}{\partial_z} = 0$
- $u = u(x, y, z)$ and $u = 0$ on the boundary (no-slip)
- velocity: Dirichlet condition in inflow
  pressure: $p - \sigma = p_{out}$ in outflow

Main point: we discretize only in x direction and we use a high polynomial degree in y and z directions to determine the profile (it is no longer parabolic)



Pipe with rectangular section, for a non newtonian fluid

OpenFoam comparison

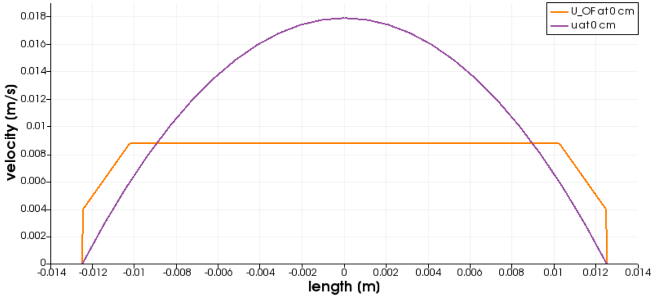


Geometry:

$H_{\text{in}} = 2.5\,\text{cm} \qquad L = 10\,\text{cm}$

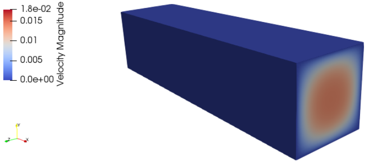
BC inlet:

- our program
  $u = \text{parabolic}$
- Open Foam
  $u = 0.0088\,\text{m}\,\text{s}^{-1}$



Pipe with rectangular section, for a non newtonian fluid

OpenFoam comparison

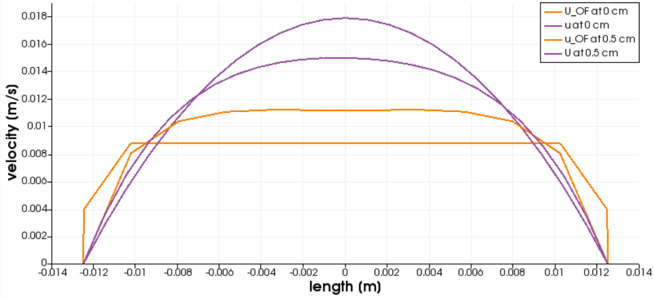


Geometry:

$H_{in} = 2.5\,\mathrm{cm} \qquad L = 10\,\mathrm{cm}$

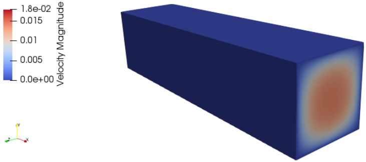
BC inlet:

- our program
  $u = \text{parabolic}$
- Open Foam
  $u = 0.0088\,\mathrm{m\,s^{-1}}$



Pipe with rectangular section, for a non newtonian fluid

OpenFoam comparison

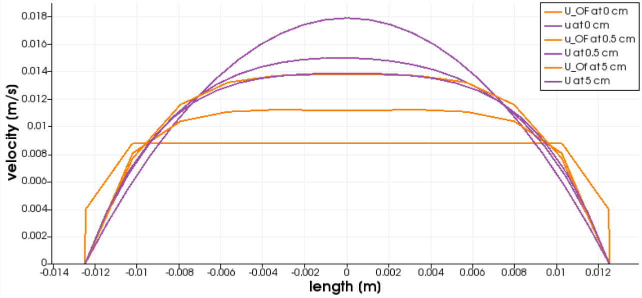


Geometry:

$H_{\text{in}} = 2.5\,\text{cm} \qquad L = 10\,\text{cm}$

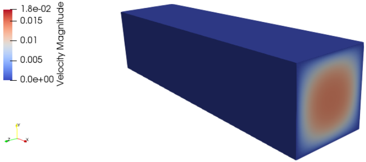
BC inlet:

- our program
  $u = \text{parabolic}$
- Open Foam
  $u = 0.0088\,\text{m}\,\text{s}^{-1}$



Pipe with rectangular section, for a non newtonian fluid

OpenFoam comparison

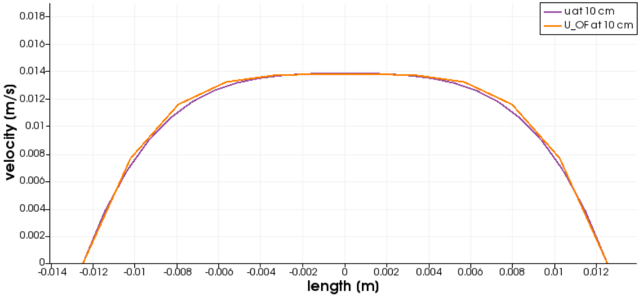


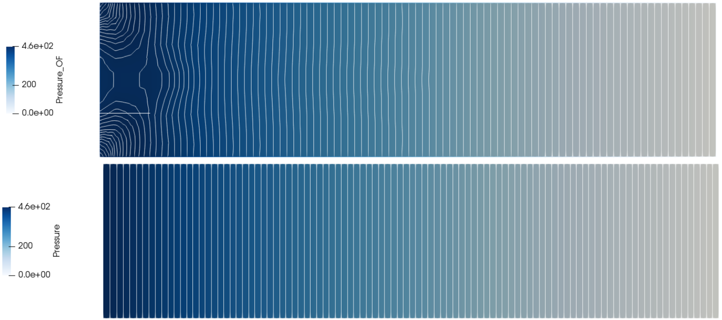
Geometry:

$H_{in} = 2.5\,\text{cm} \qquad L = 10\,\text{cm}$

BC inlet:

- our program
  $u = \text{parabolic}$
- Open Foam
  $u = 0.0088\,\text{m}\,\text{s}^{-1}$





Comparing the average piston pressure, obtained with OpenFoam, there is a value 3 % higher than that obtained with our model.

In a simulation with 80 cells, the CPU time for

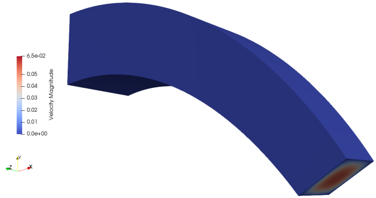
- OpenFoam is 59 s
- our program is 29 s

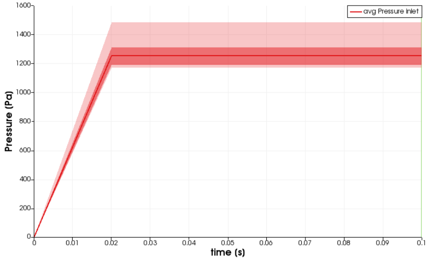
with a reduction of 50 %.

Curved pipe with variable radius, for a non newtonian fluid

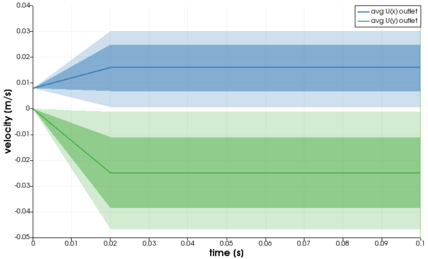
OpenFoam comparison

Geometry: $H_{\text{in}} = 2.5\,\text{cm}, \quad H_{\text{out}} = 1.25\,\text{cm}, \quad L = 10\,\text{cm}, \quad \theta = \pi/3$





The average pressure, estimated by our program, is 7 % lower than the average value on the inlet face for OpenFoam. Instead the axial velocity is 4 % higher.



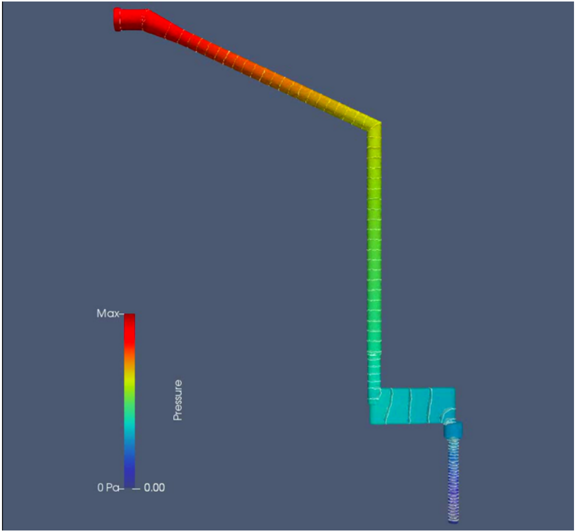
With 160 cells, the CPU time for

- OpenFoam is 8.35 min
- our program is 1.49 min

with a reduction of about 82 %.

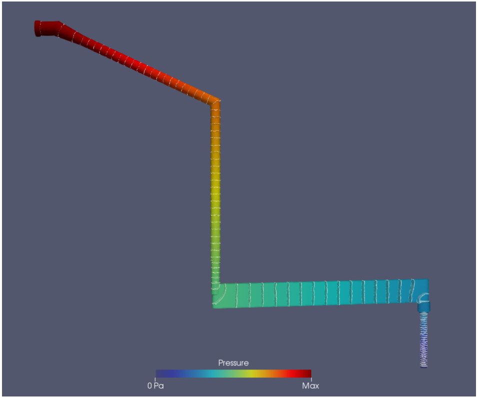
Example

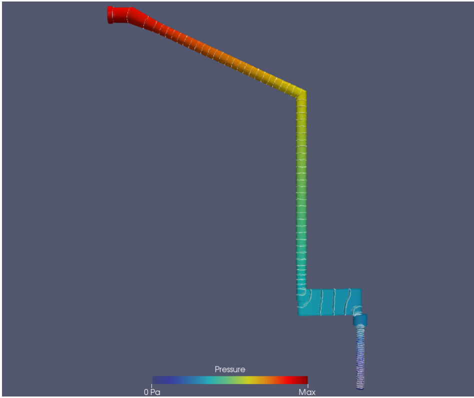
We simulate the behavior of a fluid being pushed into a channel by a moving wall. In the first 0.05 seconds the velocity is increased and then it is kept constant.



Pipes comparison

Comparison between the pressure drops in two channels which differ only in the horizontal part.





The a priori simulation of the behavior of fluids is particularly useful in the design phase because it allows to create components that respect particular physical constraints.