Multi-scale numerical modelling of debris flow: coupling 2D and 3D simulation strategies

A. Pasqua*, A. Leonardi, M. Pirulli

*andrea.pasqua@polito.it

Flow chart of the research

Motivations

- Very **fast** phenomena;
- **impossible** evacuation;
- Cause of **economic and human** loss.

Modelling tools

- **RASH3D** (shallow-water method);
- **Hybird** (3D method);
- **Less expensive** than full scale experiment;
- Computational power **almost** satisfying.

Issues

- Velocity and pressure **approximated** to the mean (RASH3D);
- Large computational time necessary for big area simulated (Hybird).

Coupling

- **Enhancement** of time calculation;
- **Reliable** results;
- **Fewer data** to process;
- **Fewer results** to interpret.

Andrea Pasqua, Alessandro Leonardi, Marina Pirulli

Question: why do we need to understand and model debris flow phenomena?

Answer: see next slide

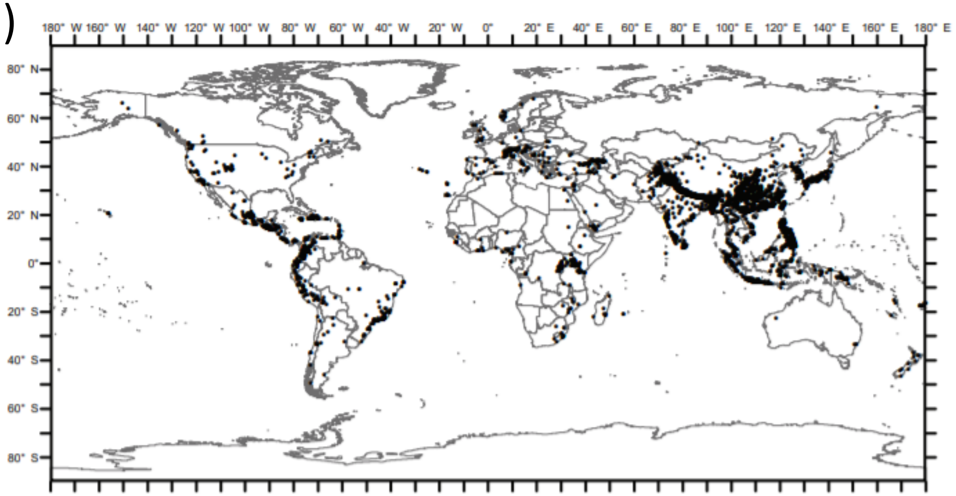
Andrea Pasqua, Alessandro Leonardi, Marina Pirulli

EGU General Assembly 2020

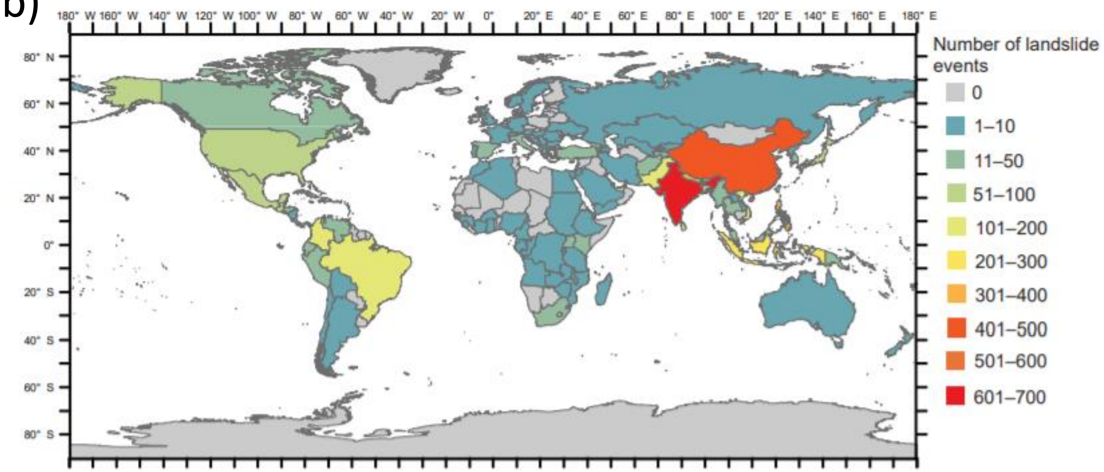
POLITECNICO DI TORINO

Motivations

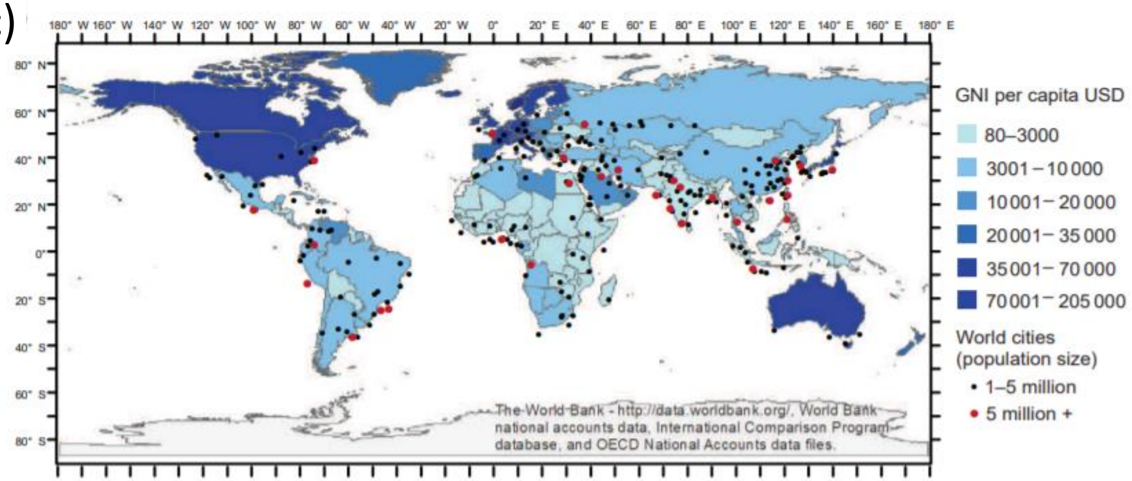
(a)



(b)



(c)



(a) The location of non-seismically triggered fatal landslide events from 2004 to 2016. Individual landslide events shown by a black dot. *

(b) (b) Number of non-seismically triggered fatal landslide events from 2004 to 2016 by country. *

(c) (c) The gross national income per capita (USD) by country (World Bank, 2018a), and the location of major urban centres globally (ESRI, 2018).*

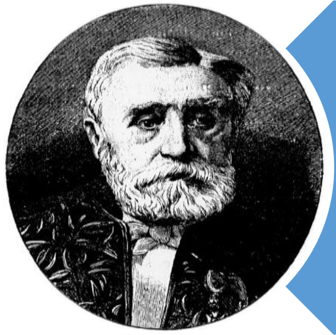
* All the images in this slides are from: **Froude & Petley (2018) Natural Hazards and Earth System Sciences, 18(8), 2161–2181.**

Question: what kind of approach are suitable for this phenomena?

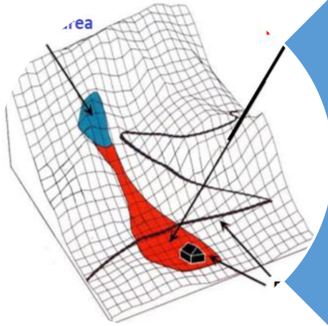
Answer: see next slide

Modelling tools

RASH3D (shallow-water model)



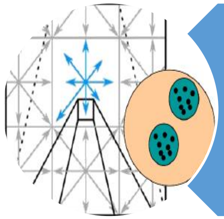
de Saint-Venant equations;



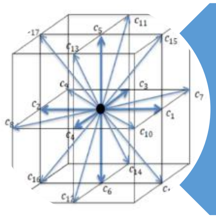
Finite volume method (FVM);

Hybird (3D model)

Lattice-Boltzmann equations



Mesoscopic method;



Collision and streaming processes;

Question: what are the limits and the issues of these approaches?

Answer: see next slide

POLITECNICO DI TORINO

Limits and issues

RASH3D (shallow-water model)

Pros:

- **Fast** analysis;
- **Reliable** results;
- Adaptable to **topography**.

Cons:

- **Mean** results;
- The run out **must** be **greater** than the flow's height
- Mitigation structure are **difficult** to **design** using this approach

Hybird (3D model)

Pros:

- Able to simulate **complex geometry**;
- Results represented **point by point**;
- **Easy to pair** with other models (DEM)
- **Allows** to **design** mitigation structures

Cons:

- Large analysis require a **large time** to run
- Results may be complex to **interpret**

Question: How can we improve the analysis? How can we have even more suitable results (in relation to informatics and graphic devices)

Answer: Coupling

Coupling

Idea:

- **Combining** the pros of each modelling tool.
- **Fewer** data to **process**
- **Fewer** results to **interpret**
- **Enhancement** of time calculation

Hypothesis:

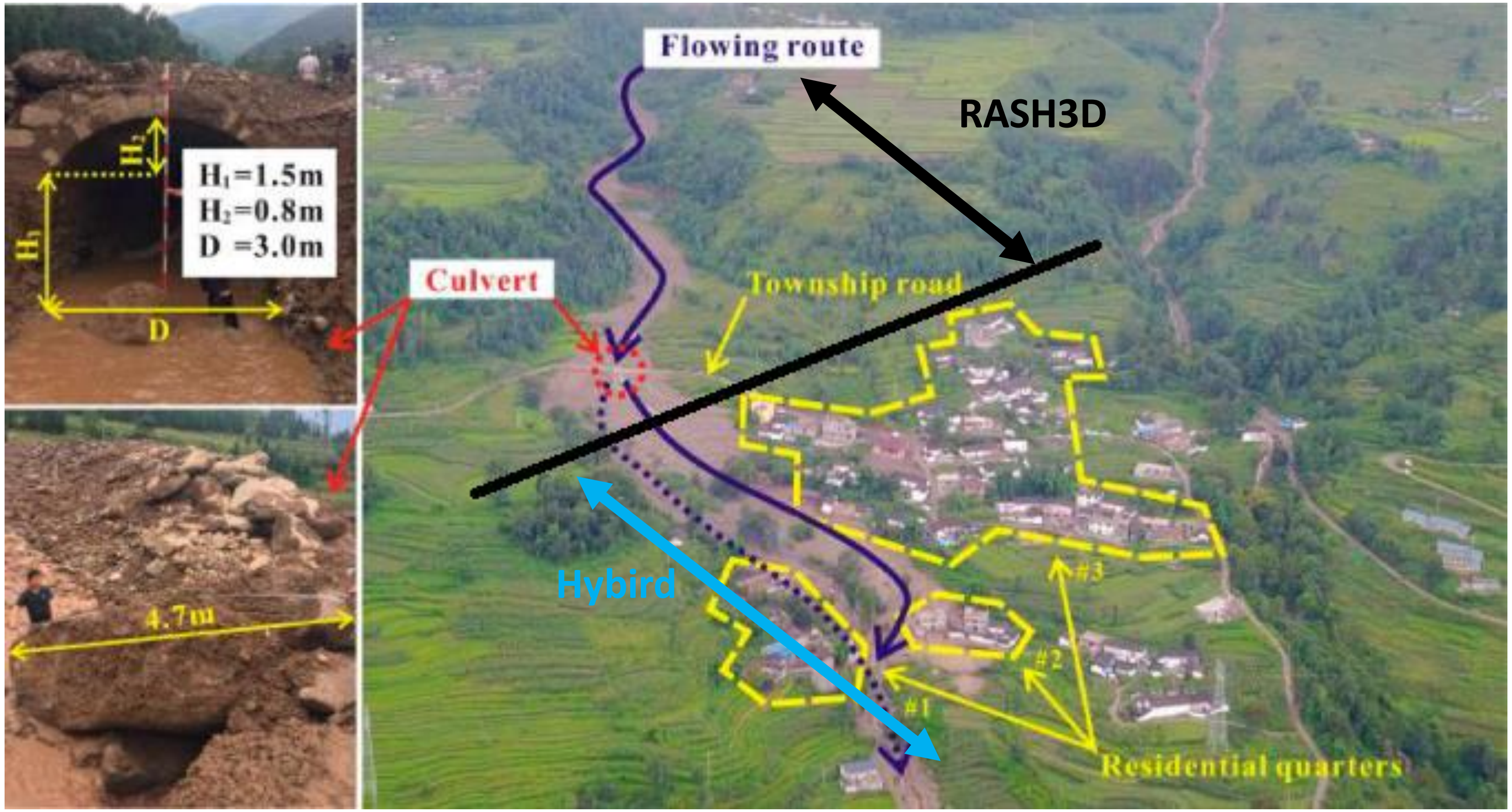
- **Dam break** problem;
- Run out **longer** than 1km;
- RASH3D + Hybird approach.

Conclusions:

- Time calculation **minimized**;
- **No loss** of information (velocity and height of the flow, forces);
- Design of mitigation structure is possible

POLITECNICO DI TORINO

Coupling

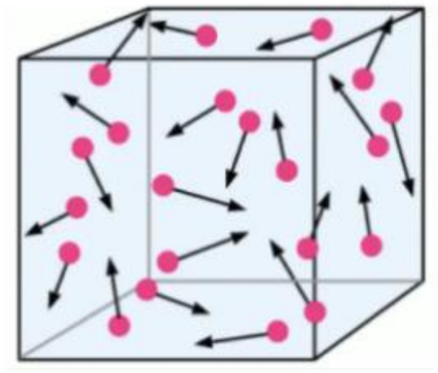


Imagine from Ming-liang Chen et al (2019), Contribution of Excessive Supply of Solid Material to a Runoff-Generated Debris Flow during Its Routing Along a Gully and Its Impact on the Downstream Village with Blockage Effects, MDPI

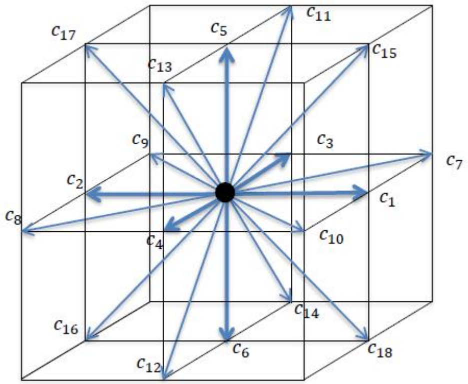
The image shows the main idea. In order to **save computational time** is useful to use an **integrated model** where it is **not necessary** to collect a large amount of information. Otherwise, if the flow is close to **strategic points**, it is necessary to have **more information** (velocity, forces, etc..) to **design** correctly the defence structures.

Focus on Lattice-Boltzmann method

Lattice-Boltzmann method considers a **set of particles** in a volume of control. The space must be discretised



The **main parameter** is the function f(**x**,t,**c**), which indicates the position (x,y,z) of a particle at the time t and what velocity that particle has (**c**).



Main stages of calculation: **streaming** and **collision**

- **Streaming**: particles' **movement** in the volume of control;
- **collision**: particles collide between them (simulation of the fluid's **viscosity**)

Continues in the next slide

Focus on Lattice-Boltzmann method

Reconstruction of macroscopic variables (density ρ and velocity **u**):

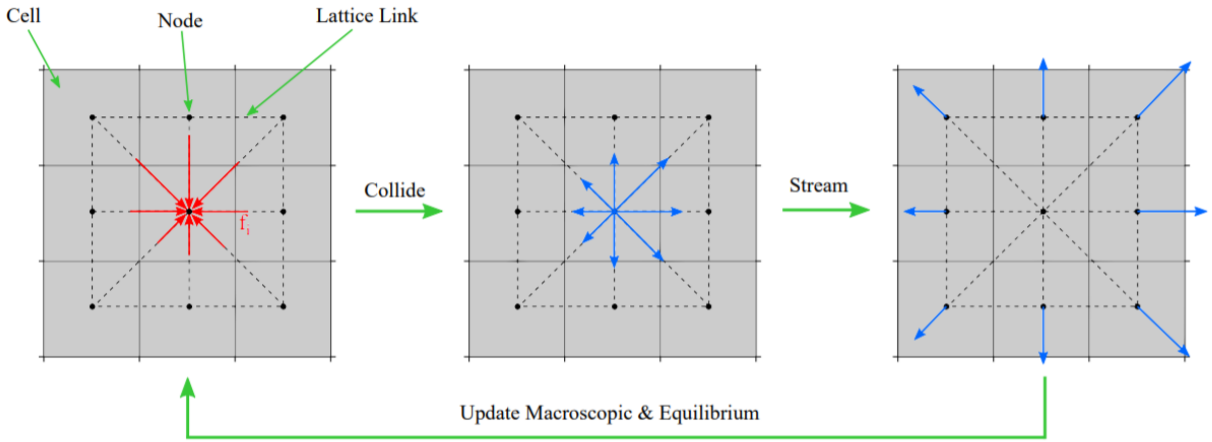
- $\rho(\boldsymbol{x},t) = \sum_i f_i$
- $u(\boldsymbol{x},t) = \frac{\sum_i f_i c_i}{\rho}$

Streaming step:

- Each particle moves into the domain due to its own velocity **c**
- Free-surface is obtain through volume of fluid method (VOF)

Collision step:

- macroscopic variables are modified by the collision between the particles



* Imagine from Adrian R. et al., A real-time modelling and simulation platform for virtual engineering design and analysis (2018), 6th European Conference on Computational Mechanics (ECCM 6) 11–15 June 2018, Glasgow, UK

Preliminary study (LBM D3Q19)

Idea: comparison between the **analytical solution** of Navier-Stokes equations (NSE) and the **numerical results**

Newtonian fluid — Results of NSE in **permanent** and **uniform** motion

Bagnold fluid — Results of NSE in **permanent** and **uniform** motion

Preliminary study (LBM D3Q19)

- Laminar flow (**Poiseuille**)
- Procedure for a **Newtonian** fluid to obtain analytical solutions:

NSE

Solutions in uniform and permanent motion (velocity and pressure)

Flow rate and height's flow

Froude number calculation

Numerical model LBM D3Q19 must verify:
- Flow rate
- Height
- Fr

If Fr>1 supercritical flow; if Fr<1 subcritical flow

Preliminary study (LBM D3Q19)

- Laminar flow (**Poiseuille**)
- Procedure for a **Bagnold** fluid to obtain analytical solutions:

1. Velocity and pressure in Bagnold fluid
2. Flow rate calculation
3. flow's height calculation
4. Froude number calculation
5. Numerical model LBM D3Q19 must verify:
   - Flow rate
   - Height
   - Fr
6. If Fr>1 supercritical flow; if Fr<1 subcritical flow

Preliminary study (LBM D3Q19)

In this presentation are studied two different **problems** which are particularly interesting for **fluid dynamics**. Both of them are studied using the coupled method (constant velocity inlet)

Problems

- **Channel** — Preliminary results (Newtonian and Bagnold)
- **Dam-break** — Preliminary results (Newtonian and Bagnold)

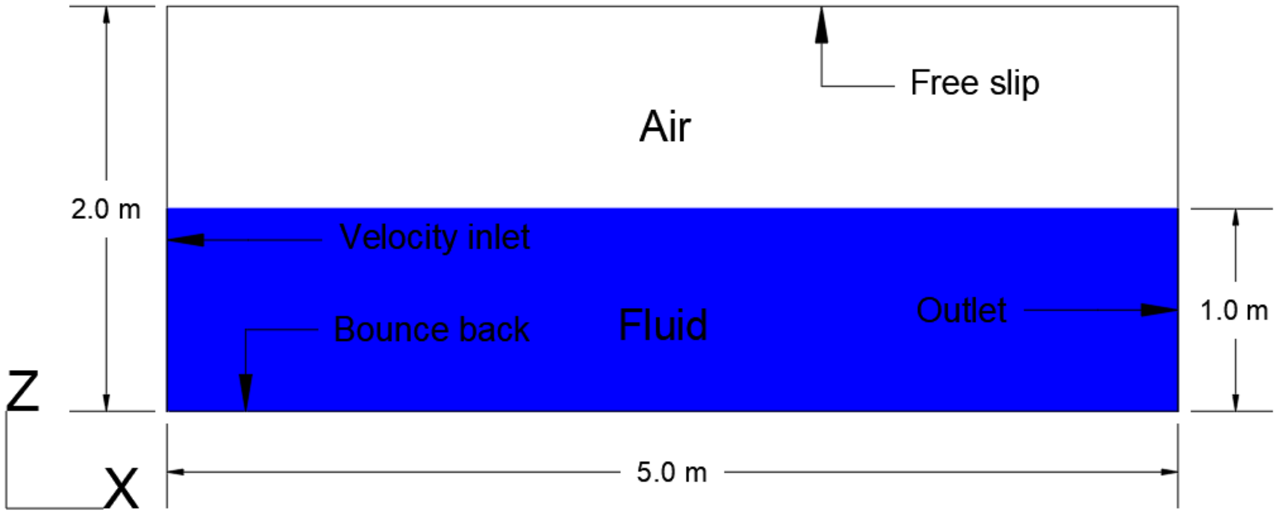
Channel

Preliminary results (Newtonian fluid)

Constant inlet velocity

- Newtonian rheology;
- BCs: **Zou-He velocity inlet** & outlet;
- fluid characteristics specified (see table);
- **Parabolic Inlet profile**

Lc [m]	5.00	**Velocity inlet [m/s]**	0.50
Hc [m]	2.00	**Velocity outlet [m/s]**	0.50
Fluid Min X [m]	0.00	**Force X [m/s^2]**	3.35
Fluid Max X [m]	5.00	**Force Z [m/s^2]**	-9.21
Fluid Min Z [m]	0.00	**ρ [kg/m^3]**	1500.00
Fluid Max Z [m]	1.00	**μ [Pas]**	60.00



Slope:
$\Theta = 20°$

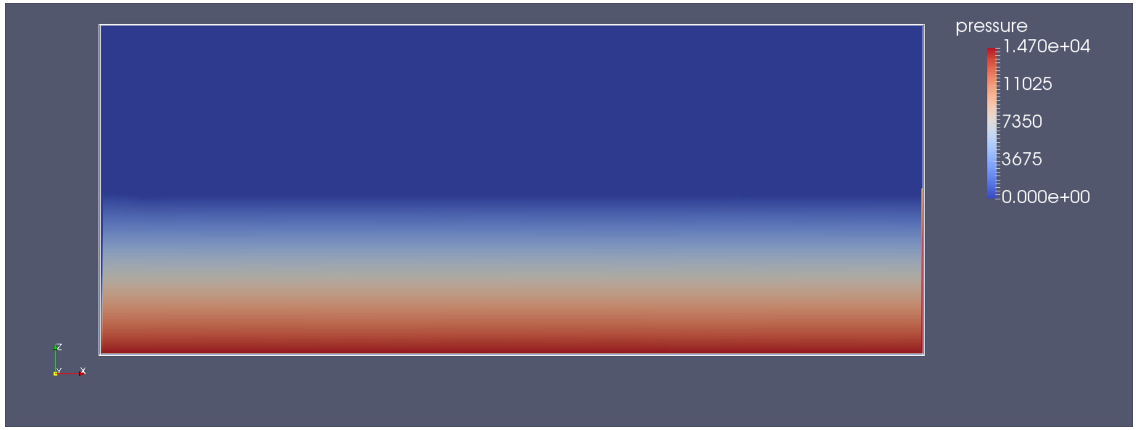
Preliminary results (Newtonian fluid)

Analytic solutions		Numeric solutions	
uMax [m/s]	0.50	**uMax [m/s]**	0.53
uMean [m/s]	0.33	**uMean [m/s]**	0.35
Fr [-]	0.1	**Fr [-]**	0.1
Re [-]	8.3	**Re [-]**	8.8

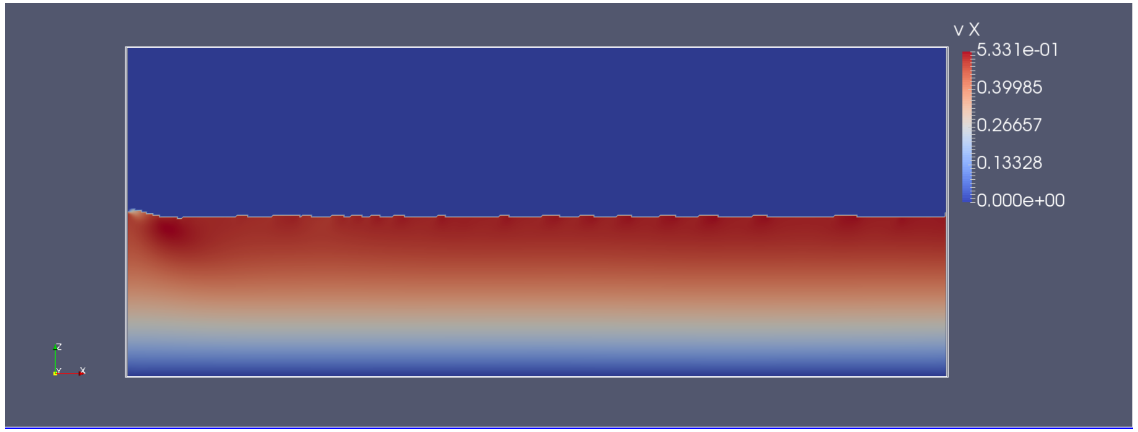
discrepancies	
uMax [m/s]	0.03
uMean [m/s]	0.02
Fr [-]	0.0
Re [-]	0.5

Numerical results

Pressure



Velocity



Preliminary results (Newtonian fluid)

- **Variable flow (height changes during the simulation)**
  - Depending on the course of the free surface, the velocity and pressure values adapt satisfactorily to the new configuration.
  - the system may be approximate to a Poiseuille flow.

Preliminary results (Newtonian fluid)

In the next slide you will able to observe how the free surface and velocity evolve during the simulation;

The **mass** is represented on the **left**

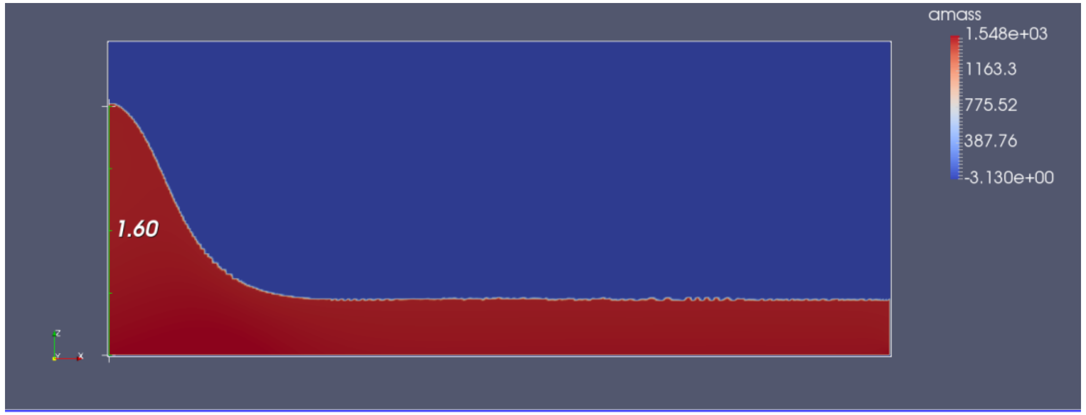
The **velocity** is represented on the **right**

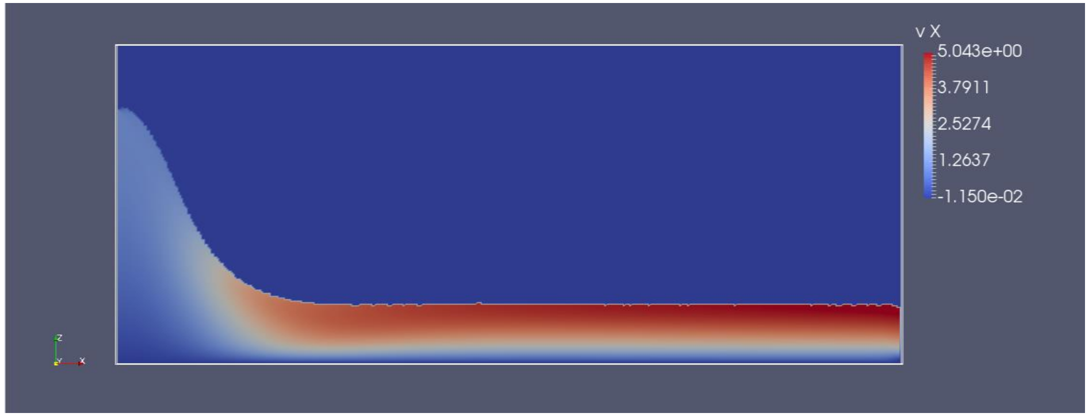
Preliminary results (Newtonian fluid)

Mass | **Velocity**

Time=0s

Time=60s





EGU General Assembly 2020

POLITECNICO DI TORINO

Dam break

Preliminary results (Newtonian fluid)

Problem

- Newtonian fluid
- Dam break

Characteristics of the fluid:

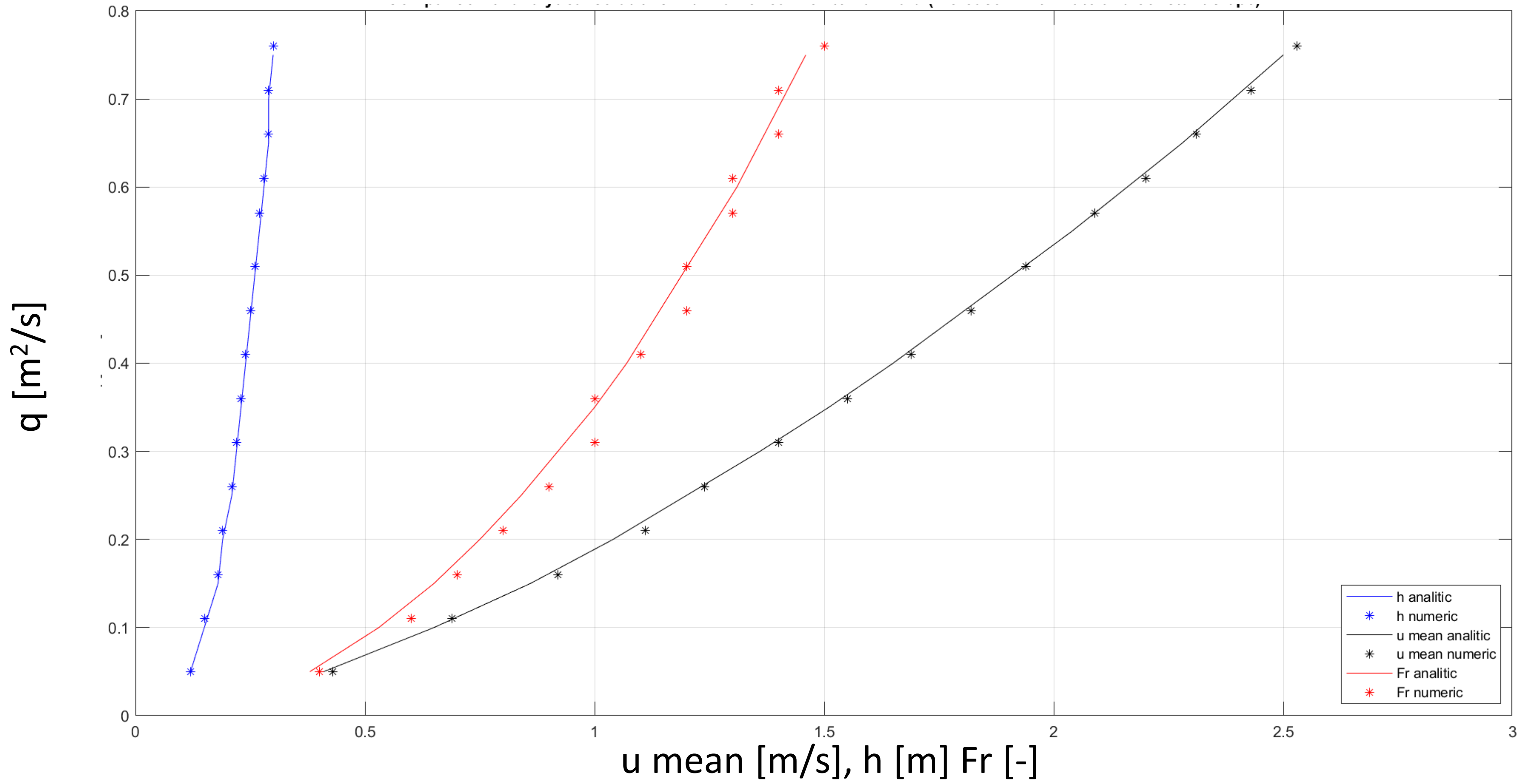
- flow's height (h) in inlet 0.50 m
- Dynamic viscosity 60.00 Pas
- Density 1500.00 kg/m^3

Problems analysed:

- Increasing of flow rate,
- Constant slope (20°)

Preliminary results (Newtonian fluid)

Comparison of analytical with numerical solutions for a Newtonian fluid (increase in flow rate and constant slope)



hInlet=0.5m

μ=60Pas

ρ=1500kg/m^3

Fx=3.35m/s^2

Preliminary results (Newtonian fluid)

Problem

- Newtonian fluid
- Dam break

Characteristics of the fluid:

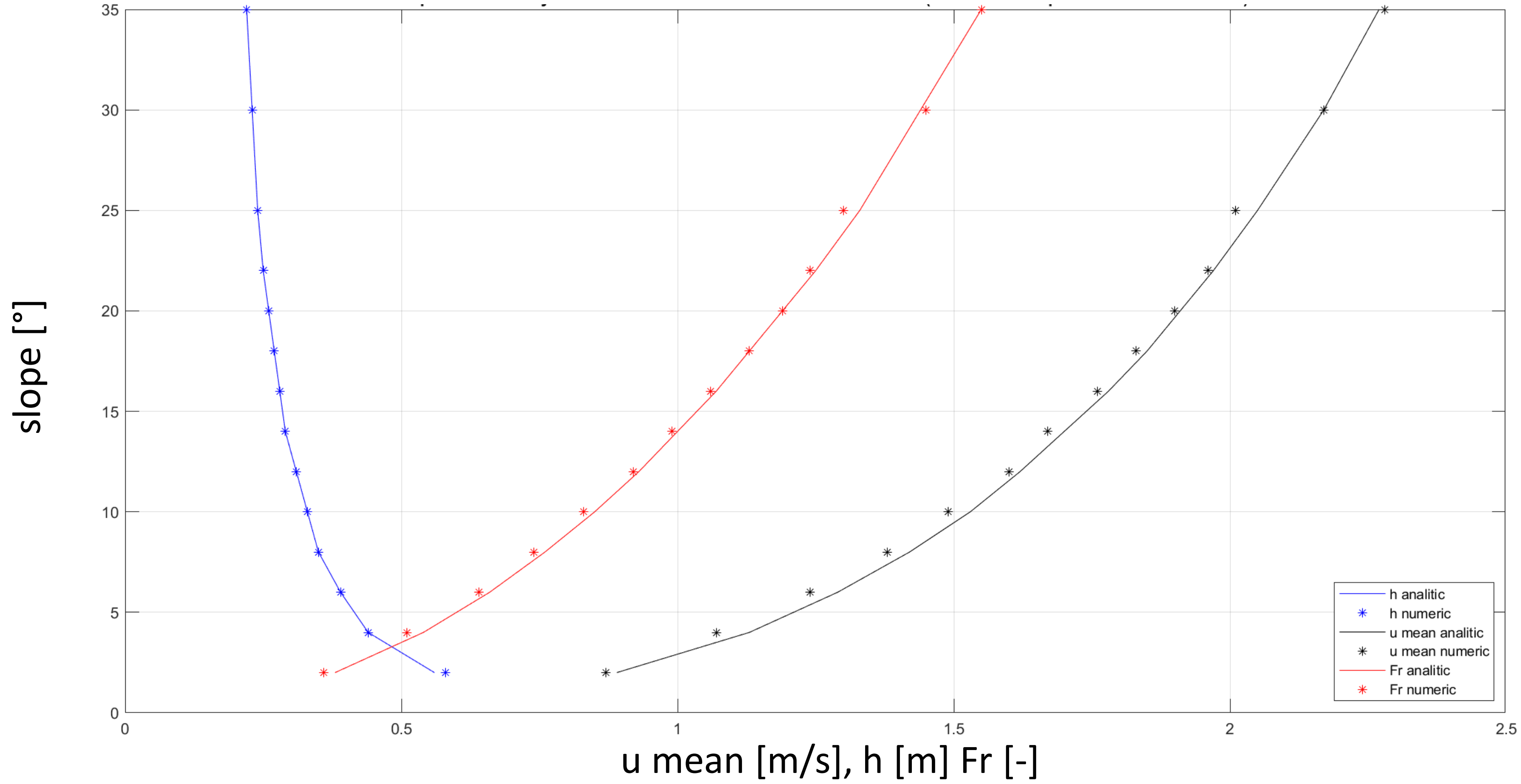
- flow’s height (h) in inlet 0.50 m
- Dynamic viscosity 60.00 Pas
- Density 1500.00 kg/m^3
- Constant flow rate

Problems analysed:

- Increasing in slope
- Constant flow rate

Preliminary results (Newtonian fluid)

Comparison of analytical with numerical solutions for a Newtonian fluid (increase in slope and constant flow rate)



hInlet=0.5m

uMeanInlet=1.0m/s

qInlet=0.5m²/s

μ=60Pas

ρ=1500kg/m³

Preliminary results (Bagnold fluid)

Problem

- Bagnold fluid
- Dam break

Characteristics of the fluid:

- flow's height (h) in inlet 0.50 m
- Dynamic viscosity 60.00 Pas
- Density 1500.00 kg/m^3

Problems analysed:

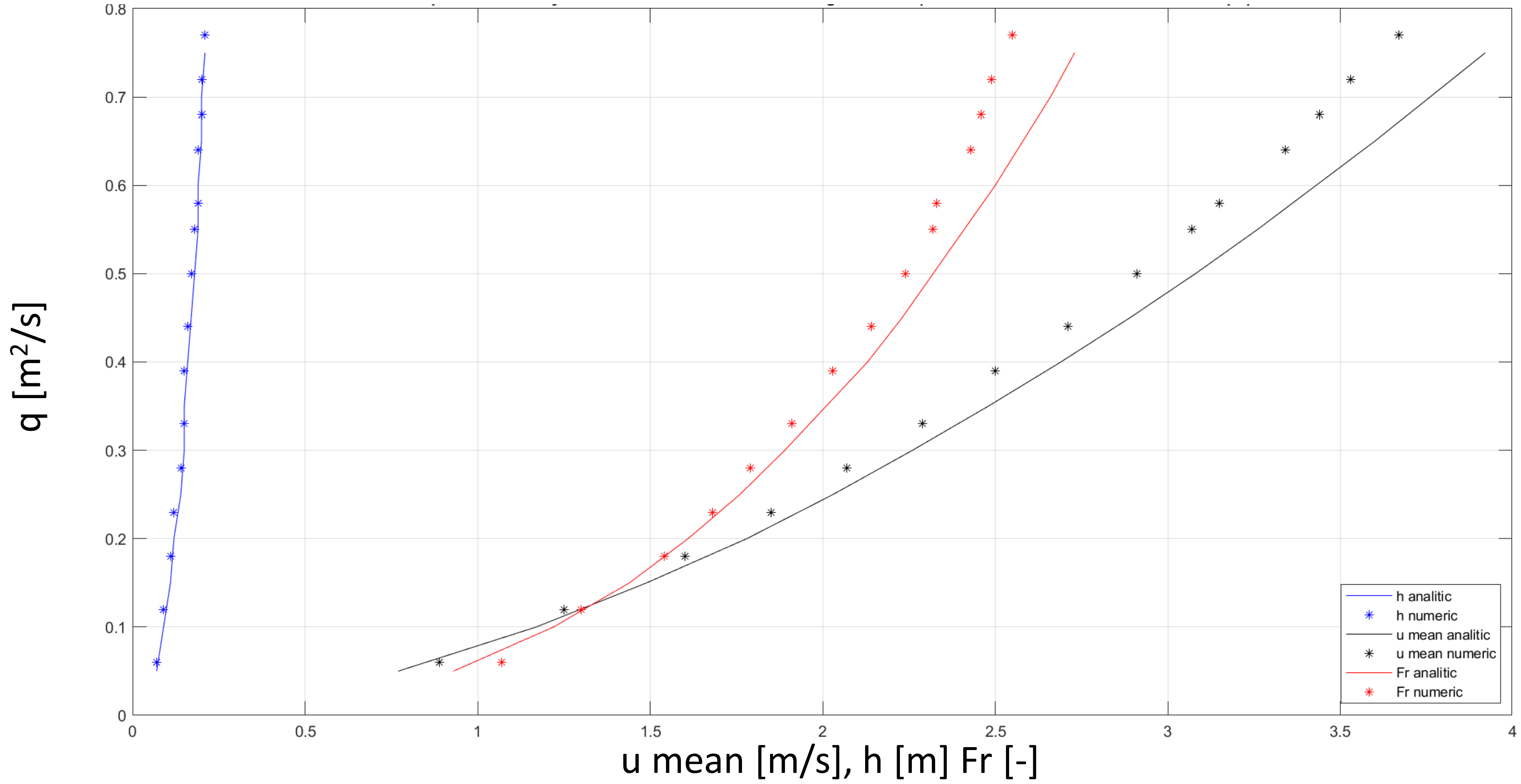
- Increasing in flow rate,
- Constant slope (20°)

EGU General Assembly 2020

POLITECNICO DI TORINO

Preliminary results (Bagnold fluid)

Comparison of analytical with numerical solutions for a Bagnold fluid (increase in flow rate and constant slope)



hInlet=0.5m

d=0.02m

ρ=1500kg/m^3

Fx=3.35m/s^2

ϑ=20°

Preliminary results (Bagnold fluid)

Problem

- Bagnold fluid
- Dam break

Characteristics of the fluid:

- flow's height (h) in inlet 0.50 m
- Dynamic viscosity 60.00 Pas
- Density 1500.00 kg/m^3

Problems analysed:

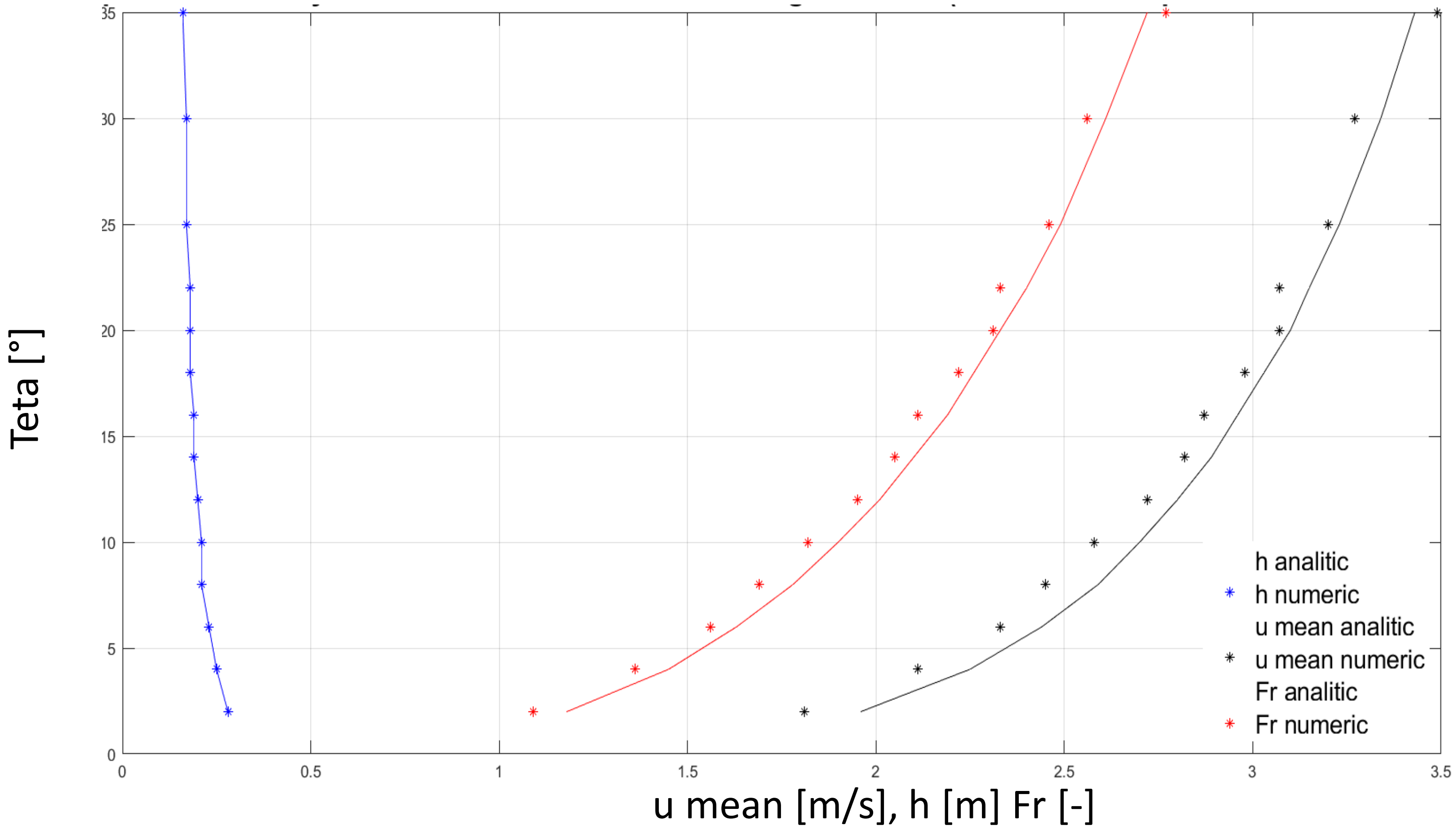
- Constant flow rate,
- Increasing in slope

EGU General Assembly 2020

POLITECNICO DI TORINO

Preliminary results (Bagnold fluid)

Comparison of analytical with numerical solutions for a Bagnold fluid (increase in slope and constant flow rate)



hInlet=0.5m

uMeanInlet=1.0m/s

qInlet=0.5m^2/s

d=0.02m

ρ=1500kg/m^3

Contacts

Andrea Pasqua
Ph.D. Student,
Dept. of Structural, Building
and Geotechnical Engineering
Politecnico di Torino
Corso Duca degli Abruzzi 24,

10124 Torino (Italy)
Mail andrea.pasqua@polito.it

Tel. +390110904914