Quantification of the vulnerability of buildings exposed to the risk of debris flows and flash floods through numerical modelling (Quorum Project CAP2025 Region)

Rime Chehade (1), Bastien Chevalier (1), Pierre Breul (1), Fabian Dedecker (2), Jean-Claude Thouret (3)
(1) Institut Pascal (2) Itasca Consultants, SAS (3) Laboratoire Magmas et Volcans

Itasca symposium

19 February 2020
Plan

1. Background
2. Numerical approach
3. Coupling PFC with fluid?
4. Parametric study
5. Comparison with empirical methods
6. Conclusions and outlooks
1. Background

- As a result of global warming, mudslides and flash floods (lahars) are increasingly frequent in the latitudes and expose urbanized areas to a significant risk.

Lahar = mixture of sediments and water originating from volcanoes.
1. Background

• **Case study: Arequipa-Peru**

  - **Location:** 17 km of the summit of the El Misti volcano
  - **Hazard:** high precipitation + volcanic ash deposits
  - **Consequences:** exposed residential areas, poor populations; infra. exposed

→ It is necessary to assess and map the risk through vulnerability quantification

![Lahars in Arequipa](image1)

Very high risk for the safety of the population
2. Numerical approach

- **General approach**

<table>
<thead>
<tr>
<th>Step 1. Modeling the lahar</th>
<th>Step 2. Modeling the interaction between lahars and structures</th>
<th>Step 3. Assessing the vulnerability of structures</th>
</tr>
</thead>
<tbody>
<tr>
<td>Blocks modeled with DEM (PFC3D)</td>
<td>Pressure induced by the flow on different types of obstacles</td>
<td>Flow vs. Structure</td>
</tr>
<tr>
<td>Fluid modeling</td>
<td>Lahars in a canal with a recess</td>
<td>Stress levels calculated from the total flow pressure</td>
</tr>
<tr>
<td>Lahars in a canal with an obstacle</td>
<td>Lahars in a canal with a recess</td>
<td>Deformation/damage to quantify the vulnerability of the structure</td>
</tr>
</tbody>
</table>
2. Numerical approach

• Modeling scales

✓ Models to obtain trajectory of the flow and global flow characteristics

Existing models

Experimental

Numerical

✓ Modeling at the scale of the structure

- Complement existing models at smaller scale
- Export suitable flow parameters to this area (h, v, ρ)
- Calculate the impact of the blocks on the structures in this region
2. Numerical approach

Lahars modeling

- Type of flow: DF or HCF
- Macro
- Parameters: h, v, ρ

- Blocks:
  - Shape
  - Granulo...

- Fluid phase: (water+fine)
  - Coupling blocs-fluide

- Geometry of the simulated channel:
  - linear
  - obstacle
  - recess...

DF

HCF

Fluid+blocks
2. Numerical approach

- Problem modeling
  
  *Solid fraction*
  
  - Blocks
  
  *Fluid phase*
  
  - Fines
  - Water

**Modeled explicitly with DEM**

- Blocks with a given size distribution

**Modeling of the fluid and its effects on blocks**

- Buoyancy effect
- Drag force
2. Numerical approach

- Model procedure

1. Representative Elementary Volume (REV) (5 m × 10 m × 1.5 m)

2. Generation of a rectangular channel (25 m × 10 m × 4 m)

Fluid velocity assumed to be constant and equal to 3 m/s
→ fixed velocity for the blocks at the entrance of the channel then released

Channel flow generation

\[ V = 3 \text{ m/s} \]
2. Numerical approach

- **Model calibration**

<table>
<thead>
<tr>
<th>Property</th>
<th>Blocks density (kg/m$^3$)</th>
<th>Fluid density (kg/m$^3$)</th>
<th>Dynamic viscosity (Pa.s)</th>
<th>Friction ball-ball</th>
<th>Friction Ball-wall</th>
<th>Rolling resistance</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density range</td>
<td>Between 2500 et 2700</td>
<td>Between 1000 et 2000</td>
<td>Between 0.03 et 0.075</td>
<td>Between 0.05 et 0.4</td>
<td>Between 0 et 1</td>
<td>Between 0 et 0.6</td>
</tr>
</tbody>
</table>

**DF characteristics searched**

- Solid concentration = 50 %
- Apparent density = 1800 kg/m$^3$
- Flow rate = 35-40 m$^3$/s

**Property calibration**

- Blocks density = 2500 kg/m$^3$
- Fluid density = 1100 kg/m$^3$
- Dynamic viscosity = 0.048 Pa.s
- Friction Ball-Ball = 0.4
- Friction wall-ball = 0.5
- Rolling resistance = 0.2

**Result**

- Solid concentration 55 %
- Apparent density 1867 kg/m$^3$
- Flow rate 38-40 m$^3$/s

---

### Iverson (1997)

#### Solid Grain Properties

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass density, kg/m$^3$</td>
<td>$\rho_s$</td>
</tr>
<tr>
<td>Mean diameter, m</td>
<td>$d$</td>
</tr>
<tr>
<td>Friction angle, deg</td>
<td>$\phi_f$</td>
</tr>
<tr>
<td>Restitution coefficient</td>
<td>$e$</td>
</tr>
</tbody>
</table>

#### Pore Fluid Properties

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass density, kg/m$^3$</td>
<td>$\rho_f$</td>
</tr>
<tr>
<td>Viscosity, Pa.s</td>
<td>$\mu$</td>
</tr>
</tbody>
</table>

#### Mixture Properties

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Solid volume fraction</td>
<td>$\nu_s$</td>
</tr>
<tr>
<td>Fluid volume fraction</td>
<td>$\nu_f$</td>
</tr>
<tr>
<td>Hydraulic permeability, m$^2$</td>
<td>$k$</td>
</tr>
<tr>
<td>Hydraulic conductivity, m/s</td>
<td>$K$</td>
</tr>
<tr>
<td>Compressive stiffness, Pa</td>
<td>$E$</td>
</tr>
<tr>
<td>Friction angle, deg</td>
<td>$\phi$</td>
</tr>
</tbody>
</table>

#### Properties

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Solid concentration</td>
<td>0.5</td>
</tr>
<tr>
<td>Apparent density</td>
<td>1800 kg/m$^3$</td>
</tr>
<tr>
<td>Flow rate</td>
<td>35-40 m$^3$/s</td>
</tr>
</tbody>
</table>

---

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Solid concentration</td>
<td>0.5</td>
</tr>
<tr>
<td>Apparent density</td>
<td>1867 kg/m$^3$</td>
</tr>
<tr>
<td>Flow rate</td>
<td>38-40 m$^3$/s</td>
</tr>
</tbody>
</table>
3. Coupling PFC with fluid?

- Geometry of the simulated flow

- Straight rectangular channel
  - Constant velocity
  - Homogeneous and constant flow field

- Complex channel
  - Choosing a code to model the fluid phase

  Solution 1: Darcy (with FLAC3D)

  Solution 2: Navier stokes FE (Telemac3D)
3. Coupling *PFC* with fluid?

- **Solution 1: coupling with a Darcy flow**

  Using *FLAC3D* flow model (*incompressible flow in a porous media*)

  Obtaining the norm and orientation of the fluid velocity vectors in each cell of the *FLAC3D* mesh

  Applying the velocity fields to the particles located in each cell (drag)

---

**Channel geometry**

**FLAC3D velocity field**

**Force history**

Flow direction

- 1
- 2
- 3
- 4
- 5
- 5m
- 2m
3. Coupling *PFC with fluid?*

**Why Darcy?**
- Easy to couple *FLAC3D* with *PFC3D*
- Two ways coupling
- Access to fluid velocity vectors in each cell of the channel

**Darcy Limitations**
- Difficult to model the free surface
- No turbulences
- Difficulties to obtain realistic velocity vectors

**Solution 2 : Simulate fluid calculation with Navier stokes FE (*Telemac3D*)**

**Objectif**
- Obtaining a velocity field of a free surface flow with defined boundary conditions
- Simple turbulence model
- Method for resolving velocity vectors and fluid depth

**Limitations**
- Simulate newtonian fluid (water)
- One way coupling
3. Coupling PFC with fluid?

- **Solution 2: Telemac3D**

  **Fluid calculation Telemac 3D:**
  
  - Obtain a flow whose average velocity and fluid height is fixed at a certain distance from the obstacle.
  - Calibration of the BC and the slope to obtain the desired characteristics of the flow.

  **Coupling steps:**
  
  1. Fluid calculation in Telemac3D to obtain the velocity vectors and the free surface height
  2. Analysis and verification of results
  3. Exporting the CFD file to PFC3D

  The "CFD" file contains (X, Y, Z, Vx, Vy, Vz, Hfluid)
4. Parametric study

- **Code Telemac3D / influence of h, v and ρ s**

<table>
<thead>
<tr>
<th>Case</th>
<th>Flow height h</th>
<th>Nb Froude</th>
</tr>
</thead>
<tbody>
<tr>
<td>ref case</td>
<td>1.5 m</td>
<td>0.79</td>
</tr>
<tr>
<td>Case 2</td>
<td>3 m</td>
<td>0.55</td>
</tr>
<tr>
<td>Case 3</td>
<td>4 m</td>
<td>0.5</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Case</th>
<th>Velocity v</th>
<th>Nd Froude</th>
</tr>
</thead>
<tbody>
<tr>
<td>ref case</td>
<td>3 m/s</td>
<td>0.79</td>
</tr>
<tr>
<td>Case 2</td>
<td>4.5 m/s</td>
<td>1.18</td>
</tr>
<tr>
<td>Case 3</td>
<td>6 m/s</td>
<td>1.56</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Case</th>
<th>Blocks density ρ s</th>
</tr>
</thead>
<tbody>
<tr>
<td>ref case</td>
<td>2500 kg/m³</td>
</tr>
<tr>
<td>Case 2</td>
<td>2700 kg/m³</td>
</tr>
<tr>
<td>Case 3</td>
<td>3000 kg/m³</td>
</tr>
</tbody>
</table>

![Obstacle 1x1x8](image)
4. Parametric study

- **Influence of the flow parameters (h, v and \( \rho_s \)) on the impact pressure of blocks on a structure**

- When the flow height is higher, it is noticed that the pressure of the blocks on the obstacle is higher.
- The impact of the blocks on the obstacle is much greater with a higher flow velocity.
- Density influences the result of the impact but not strong enough as the two other parameters h and v.
5. Comparison with empirical models

- References

Zeng et al. 2015: Models of impact force of large boulders mixed in debris flow

Hubl et al. 2010:

\[
p_{\text{max}} = 5 \cdot \rho_{\text{M1}} \cdot v^{0.8} \cdot (g \cdot h_{\text{M1}})^{0.6}
\]

Table 1. Debris flow properties estimated on each event using a combination of computed impact forces, empirical factors \(k\) and \(a\), and the Froude-number

<table>
<thead>
<tr>
<th>Torrent</th>
<th>bM1 [m]</th>
<th>(\rho_{\text{M1}}) [kg/m³]</th>
<th>(v) [m/s]</th>
<th>(P_{\text{max}}) in MN/m²</th>
<th>(k)</th>
<th>(a)</th>
<th>(F_{\text{r}})</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rio Reventado</td>
<td>8.12</td>
<td>1130-1980</td>
<td>2.9-10</td>
<td>0.7</td>
<td>4.67</td>
<td>18.67</td>
<td>0.50</td>
</tr>
<tr>
<td>Humchuit Gully</td>
<td>3.5</td>
<td>2000-2300</td>
<td>10-12</td>
<td>0.7</td>
<td>8.33</td>
<td>2.31</td>
<td>1.90</td>
</tr>
<tr>
<td>Bullock Creek</td>
<td>1.0</td>
<td>1950-2130</td>
<td>2.5-5.0</td>
<td>0.13</td>
<td>6.50</td>
<td>4.06</td>
<td>1.26</td>
</tr>
<tr>
<td>Pine Creek</td>
<td>0.1-1.5</td>
<td>1970-2030</td>
<td>10-31</td>
<td>0.3</td>
<td>21.43</td>
<td>0.38</td>
<td>7.56</td>
</tr>
<tr>
<td>Wrightwood Canyon (1969)</td>
<td>1.0</td>
<td>1620-2130</td>
<td>0.6-3.8</td>
<td>0.07</td>
<td>3.68</td>
<td>4.09</td>
<td>0.95</td>
</tr>
<tr>
<td>Wrightwood Canyon (1941)</td>
<td>1.2</td>
<td>2400</td>
<td>1.2-4.4</td>
<td>0.15</td>
<td>5.21</td>
<td>6.94</td>
<td>0.87</td>
</tr>
<tr>
<td>Lesser Almatinka</td>
<td>2.0-10.4</td>
<td>2000</td>
<td>4.3</td>
<td>0.6</td>
<td>4.29</td>
<td>6.12</td>
<td>0.84</td>
</tr>
<tr>
<td>Nofri River</td>
<td>2.3-2.4</td>
<td>1810-1950</td>
<td>13.0</td>
<td>0.44</td>
<td>10.07</td>
<td>1.37</td>
<td>2.71</td>
</tr>
</tbody>
</table>

*Mean values based on the Hubl & Holzinger Formula
**Mean values

Hong et al. 2014: Statistics of the maximum impact pressure and total discharge from 1960 to 2000 in Jiangja Ravine, China

Hu et al. 2011: Hydrodynamic models, \(p\) is the impact pressure

\[
P = k\rho v \text{ or } P = k\rho v^2 \cos^2 \theta
\]

\[
p = c_m (0.5\rho gh + \rho V^2)
\]
5. Comparison with empirical models

- Model analysis - Reference case /Telemac3D

✓ Numerical model: $h_{\text{fluide}} \approx h_{\text{blocs}}$

Max blocks pressure $P_1 = 122.31$ kPa
Fluid pressure $P_2 = 17.6$ kPa

$P_{\text{total}} = P_1 + P_2 = 140$ kPa

✓ Analytical model (Hubl et al. 2010)

$$p_{\text{max}} = 5 \cdot \rho_M \cdot \psi^{0.8} \cdot (g \cdot h_M)^{0.6}$$

$P_{\text{max}} = 155$ kPa
6. Conclusions and outlooks

Conclusions

✓ Make a flow model able to reproduce desired macro characteristics

✓ Use a simplified method to model the fluid with the aim of obtaining a fluid flow field around the obstacle

✓ Measuring the forces induced by the blocks on the structures

✓ Comparison of effort results with existing analytical models

Outlooks

✓ Effect of other parameters remains to be identified e.g.: blocks size and shape, dynamic viscosity, orientation and position of obstacles..

✓ Use efforts generated by debris flows (blocks + fluid) to quantify the vulnerability of structures

✓ Analyze the damage from the flow intensity, the impact pressure on the structures by bibliography.
6. Conclusions and outlooks
Thank you for your attention

REFERENCES


Bugnion " Measurements of hillslope debris flow impact pressure on obstacles “(2011).