



CHEESE



WP2 Internal Meeting:
OpenPDAC

M. de' Michieli Vitturi, T. Esposti Ongaro,
F. Pardini, A. Neri

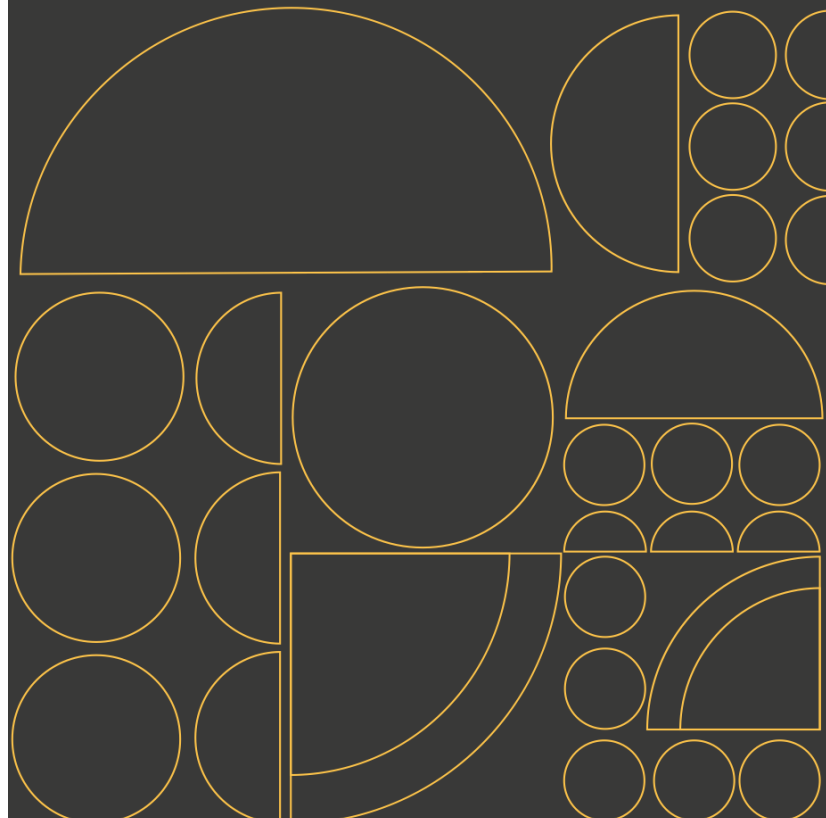
Istituto Nazionale di Geofisica e Vulcanologia,
Sezione di Pisa, ITALY



Funded by the European Union. This work has received funding from the European High Performance Computing Joint Undertaking (JU) and Spain, Italy, Iceland, Germany, Norway, France, Finland and Croatia under grant agreement No 101093038.



EuroHPC
Joint Undertaking



OpenPDAC

OpenPDAC is a parallel numerical code based on OpenFOAM (C++). It is a three-dimensional non-equilibrium Eulerian-Eulerian-Lagrangian multiphase flow model able to simulate the dynamics of small volcanic explosion and the trajectories of ballistic clasts. While the Eulerian formulation is indeed more suited for large numbers of phases of small particles, strongly coupled with the gas phase (i.e., for grains with diameter below about 1 mm), the Lagrangian one is more adapted to loosely coupled regimes (i.e., particles above a few cm).

Code repository:

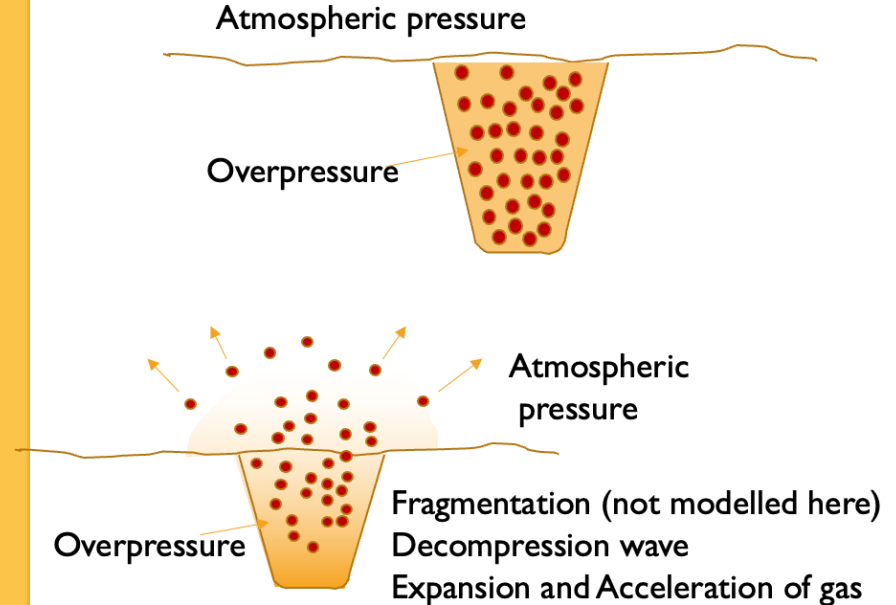
<https://github.com/demichie/OpenPDAC-10>

DOI 10.5281/zenodo.7701703

Documentation:

Work in progress...

Sketch of sudden explosive event (e.g. phreatic explosion)



OpenPDAC: Actual performances

Grid

3×10^6 cells, minimum size
1.25m

HPC

144 Cores Intel Xeon 2.3
GHz (Vesuvio, INGV-PI)

Total time

Approx. 12 hours for a
simulation of 30s

Example of Phreatic event at Vulcano Island (Italy)

Time: 22.8 s



OpenPDAC: Info for Task 2.2

01

Code release

OpenPDAC is a CPU-only (MPI) code, based on the distribution of OpenFOAM released by the OpenFOAM foundation (openfoam.org)

02

Code repository

The code repository is on Github:
<https://github.com/demichie/OpenPDAC-10>

03

Mini-apps

At present there are no mini-apps planned, but we can easily create one from a lighter OpenFOAM solver.



OpenPDAC: planned developments

01

Physics

Add new rheological models

02

Physics

Improve the modeling of the interaction between volcanic eruption mixture and water

03

Physics

Better description of the particle grain size distribution

04

Preprocessing/numerics

Use of adaptive meshes

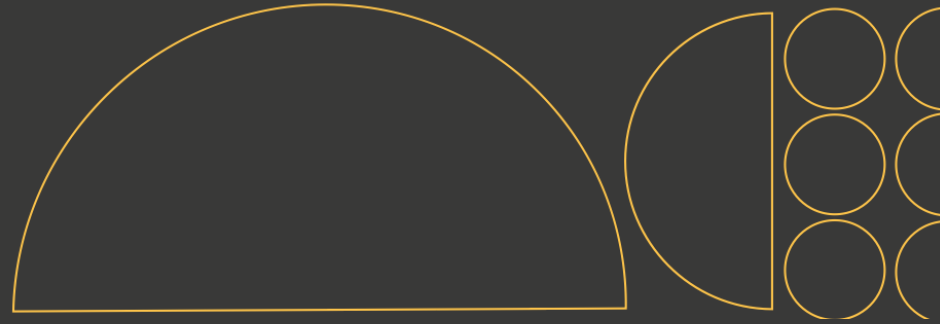
05

CI/CD

Rapid porting to future versions of OpenFOAM (available for OpenFOAM 10, soon for OpenFOAM-dev)



Thank you!



Contacts: M. de' Michieli Vitturi, T. Esposti Ongaro
Main contact: mattia.demichielivitturi@ingv.it

