Geosci. Model Dev. Discuss., 6, C1161–C1165, 2013
www.geosci-model-dev-discuss.net/6/C1161/2013/

Geoscientific Model Development Discussions

Interactive comment on "A semi-implicit, second order accurate numerical model for multiphase underexpanded volcanic jets" *by* S. Carcano et al.

S. Carcano et al.

susanna.carcano@gmail.com

Received and published: 25 July 2013

We thank Prof. Francis X. Giraldo for his helpful comments and suggestions. In order to improve the paper, we have addressed them as follows:

Question 1 - For those of us not well-versed on volcanic flow problems it is difficult to discern how this work is different from existing models. It is imperative for the authors to state what hole in the literature this paper fills. For example, are all volcanic eruption models low-order? If there are some that are high-order, how does this model differ?

The paper presents a rigorous verification of a multiphase flow model for volcanolog-

ical applications and demonstrates the need for (at least) second order numerical discretization schemes. Indeed numerical accuracy may significantly influence the simulation of two critical aspects of the fluid dynamics of a volcanic eruption: the decompression of the mixture above the crater and the description of turbulence. Since none of these phenomena had been rigorously investigated by means of non-equilibrium multiphase flow models in the context of volcanological research we have started our study from the decompression dynamics of volcanic jets. Given the large uncertainty in the available data and measurements, especially concerning the initial conditions of the flow, the use of higher order methods has never been considered a major priority of volcanological research. To the best of our knowledge, multiphase numerical models for volcanic eruptions proposed in the literature are at most of order two (Dartevelle et al., 2004 - first order; Pelanti and LeVeque, 2005 - second order; Dufek and Bergantz, 2007 - second order), but the details of spatial and temporal discretization and the effect of numerical errors on the simulated dynamics are seldom specified in the published papers.

Question 2 - P. 401, line 18: Is turbulence included in your model? If so, how is it represented in your model?

The complete PDAC model actually accounts for dissipative effects and turbulence, by means of a LES model with Smagorinsky closure (see Esposti Ongaro et al., 2007). However, in this paper we are focusing on the validation of the PDAC modelling approach for regimes corresponding to explosive volcanic eruptions. For these high Mach number flows, a scale analysis (see Carcano et al. (2012) published online) shows that turbulent effects can be neglected in a first approximation. Numerical evidence supports this analysis. Indeed, as discussed in the author comment C760, comparison of the results of the same high Mach number test case with or without the turbulence closure terms demonstrates that, in the development of the supersonic jet

above the vent, the dynamics is controlled by inertial, pressure, and drag terms.

Question 3 - P. 409, line 4: Later in the manuscript you state that you use $\theta = 1/2$ but here you do not state that. You should state so here. If so, do you use a time-filter? If not, why do the solutions remain stable. Crank-Nicholson is notoriously neutrally stable which, through nonlinear interactions, may become unstable.

Actually, in order to guarantee the stability of the time-advancing scheme, our numerical simulation have been carried out using $\theta = 0.55$. We are going to add this information at Page 409, line 2 in the manuscript, by saying:

"*In general, the value $\theta = 0.55$ is employed in most numerical simulations in order to guarantee the stability of the time-advancing scheme.* "

Question 4 - P. 410, line 1: The system is represented by a 6N system of equations. Can the problem size be reduced to a smaller system? How are the system of equations solved, iteratively? With what method and is preconditioning used?

The system consists of $6N$ equations. However, the IMF iterative solution procedure only acts on $4N$ equations (i.e. continuity and momentum equations), whereas in this stage the enthalpy equations are decoupled from the previous ones (they are only coupled by the nonlinear drag terms). The IMF iterative solver for continuity and momentum equations is based on the Newton method with an approximated Jacobian matrix. We are clarifying the solution procedure by changing

Page 410, line 9: "*the coupled continuity and momentum equations are solved*

iteratively by the approximate Newton method to update velocity fields, pressure and volumetric fractions;"

Page 410, line 12: "*the energy equations are linear in the temperatures and decoupled from the continuity and momentum equations and can finally be solved directly.* "

Question 5 - P. 416, line 1-5: What are the corresponding Courant numbers for the time-steps mentioned?

In all the simulations carried out in the present work, the maximum Courant numbers for the velocity reached in the domain are around 0.2 in the expansion region. For those values of the velocity Courant number, the maximum Courant number for the gas speed of sound is 0.1 in the simulations on the laboratory scales, whereas it becomes slightly greater than the velocity Courant number on the realistic volcanic scales. We have observed that the convergence of the algorithm becomes slow when the velocity Courant number increases over 0.2, thus we imposed this constrain in our simulations. Even if the implicit treatment of pressure gradients should allow to reach higher values of the Courant number for the speed of sound, actually we are strongly limited by the explicit treatment of the advective terms. These results have suggested us that probably a fully explicit scheme would be more efficient and robust and we are going to investigate further this alternative approach.

Question 6 - Are the 3D simulations computed serially? If not, how many processors and how long does it take? Our experiences have found that 3D Simulations for Navier-Stokes in parallel are much faster than 2D serial simulations (even with the modest number of processors, $<$ 500).

In Esposti Ongaro et al. (2007) it has been shown how the scalability of the original PDAC code was quite good on several different architectures up to 200 processors. Moreover it has been verified how the performance depends on the problem size as well as on the domain partitioning topology. However this topic has not been addressed for the improved version of the numerical model.

Both 2D and 3D simulations presented in this work are computed in parallel on a Linux cluster with 128 CPUs at 2.4 GHz with total peak performance of 580 Gflops/s. In particular 2D simulations have been carried out with a number of processors that varies between 4 and 8, depending on the resolution of the grid, whereas 3D simulations have been carried out on 32 processors. The 3D results reported at Pages 450-452 have been obtained with 1.8 millions of cells and the computation has taken 44 hours. This information will be added in the description of the numerical test case.

Interactive comment on Geosci. Model Dev. Discuss., 6, 399, 2013.