Geosci. Model Dev. Discuss., 7, C770–C780, 2014
www.geosci-model-dev-discuss.net/7/C770/2014/
© Author(s) 2014. This work is distributed under
the Creative Commons Attribute 3.0 License.

Geoscientific Model Development
Discussions
Open Access

Interactive comment on "Direct numerical simulations of particle-laden density currents with adaptive, discontinuous finite elements" *by* S. D. Parkinson et al.

S. D. Parkinson et al.

s.parkinson11@imperial.ac.uk

Received and published: 5 June 2014

The authors thank the referee for providing this review. They agree that some changes and clarification would improve the manuscript. We would propose to make the revisions outlined below for submission to Geoscientific Model Development. Each item starts with the reviewer's comment.

1 Specific comments

1. *Page 3222, line 6-9: "Small scale laboratory experiments can provide useful insight into the dynamics of these currents, but are limited by scaling issues and the available measurement techniques (Kneller and Buckee, 2000)." Is there any particular reason why model simulations are not compared directly to experiments? Validation by comparison to other model results is generally insufficient, unless you can be certain that the numerics correctly represent the underlying physical processes. More succinctly, the authors should address the question: With the advanced numerical formulation presented in this paper, has progress actually been made on representing the underlying turbidity currents? If not, it should instead be pointed out that the emphasis of this article is on providing a framework for addressing deficiencies in the current physics formulation.*

The model is directly compared to experimental data of deposit depth. However, this is the only direct comparison with experimental data and we agree that comparison with experimental results is not a major focus of this paper.

The principal focus of this paper is validation of novel computational methods for this application. For this reason we choose to compare the model against the previously published, well regarded three-dimensional DNS models of Necker et al. (2002) and Espath et al. (2013). The equations upon which this model is based, and variations upon them, are well established for modelling sediment-laden density currents of this type. They have been validated against experimental results in a number of different situations using a range of diagnostics (Sequeiros et al., 2009; Necker et al., 2002; Espath et al., 2013; Huang et al., 2007; Georgoulas et al., 2010).

The authors agree that this could be made more clear and propose changing the end of the introduction to read

GMDD
7, C770–C780, 2014

This paper presents a novel finite-element (FE) particle-laden density current model that has been built within Fluidity, an open source, general purpose, multiphase computational fluid dynamics code (Imperial College London AMCG, 2014). The model utilises adaptive meshing technologies to produce high resolution DNS models of particle-laden density currents in three dimensions, at a fraction of the cost of traditional FE models. This paper simulates a lock release density current at a Grashof number of 5×10^6 in two and three dimensions with a configuration similar to that of Necker et al. (2002). The governing equations are well established and have been validated extensively against experimental data across a range of simulation configurations (Sequeiros et al., 2009; Necker et al., 2002; Espath et al., 2013; Huang et al., 2007; Georgoulas et al., 2010). This paper validates the use of novel computational methods, including unstructured mesh adaptivity and discontinuous finite-elements, through convergence analyses and by direct comparison with the results from the previous models of Necker et al. (2002) and Espath et al. (2013), providing a framework for future modelling efforts of this type. It is shown that adaptivity reduces the required element count by approximately two orders of magnitude for this application. The computational savings afforded by adaptivity, along with the flexibility of FE discretisations and other benefits of using an unstructured adaptive mesh, will allow for simulations in complex and extended domains.

2. *Page 3230, line 26: Advection and diffusion are coupled using a first-order coupling strategy. Can you comment briefly on the validity of this approach? Is there any influence on the results by a more frequent application of diffusion?*

Diffusion is only applied once. The authors agree that the description of how this subcycling works in the manuscript is unclear and propose modifying the sentence

GMDD
7, C770–C780, 2014

describing this to clear up ambiguity

> Advection is calculated using 10 explicit subcycles with time steps of $\Delta t/10$. Diffusion/Viscous dissipation is solved using the simulation time step, Δt, and a Crank-Nicolson discretisation. This is described in detail within the Fluidity manual (Imperial College London AMCG, 2014).

For information, we have described the process in detail below.

The discretised advection-diffusion equation for a scalar, c, can be written in matrix form as

$$M\frac{c^{n+1}-c^n}{\Delta t} + A\left(u^{n+1/2}, u_s\right)c^n + Kc^{n+1/2} = 0\ , \tag{1}$$

where:

- **M** is the mass matrix,
- c^n is the known value of c from the end of the last timestep,
- c^{n+1} is the unknown value of c that we are solving for,
- $c^{n+1/2} = (c^n + c^{n+1})/2$ is the Crank-Nicolson time-discretised value of c,
- $u^{n+1/2} = (u^n + u^{n+1})/2$ is the Crank-Nicolson time-discretised velocity,
- $A\left(u^{n+1/2}, u_s\right)$ is the advection matrix, which is a function of the velocity and sinking velocity,
- K is the diffusion matrix.

GMDD
7, C770–C780, 2014

CC BY

This can be reformulated as

$$M\frac{c^{n+1}-c^*}{\Delta t}+M\frac{c^*-c^n}{\Delta t}+A\left(u^n,u^{n-1},u_s\right)c^n+Kc^{n+1/2}=0\,, \tag{2}$$

and then split into two equations as

$$M\frac{c^*-c^n}{\Delta t}+A\left(u^{n+1/2},u_s\right)c^n=0\,, \tag{3}$$

$$M\frac{c^{n+1}-c^*}{\Delta t}+Kc^{n+1/2}=0\,. \tag{4}$$

Equation (3) is subcycled in n steps with a CFL criteria one order of magnitude tighter than that implied by Δt, such that we solve n equations of the form

$$M\frac{c^\dagger-c^{\dagger-1}}{\Delta t/n}+A\left(u^{n+1/2},u_s\right)c^{\dagger-1}=0\,, \tag{5}$$

where $c^{\dagger-1}=c^n$ for the first subcycle, $c^\dagger=c^*$ at the end of the last subcycle.

3. *Page 3236, line 13-26: The effect of mesh adaptivity appears to be to add numerical diffusion to the simulation via the regridding procedure. This is likely the reason why more frequent adaptation leads to improved stability in the boundary layer. Is there any way to quantify the effect of this diffusion? In addition to explicit diffusion and upwinding, adaptive remeshing is then the third source of diffusion in the simulation.*

It is very hard to separate the diffusion introduced by the adapt process from the diffusion introduced by the slope limiting and upwinding. It is not possible to do this for the simulation in this paper. However, the Galerkin projection is used which is minimally diffusive (Farrell et al., 2009), and hence the diffusive effect of each adapt operation is small. We can also state from experience, and based on results that will hopefully form

GMDD
7, C770–C780, 2014

CC BY

part of a follow on paper, that the diffusion introduced by adaptivity and data transfer is insignificant compared to the other sources of diffusion.

Although it is possible that the small amount of diffusion introduced by adapting more frequently may have improved stability, it was clear from our analysis that the main reason adapting resulted in a more stable simulation was that small instabilities were not allowed to grow. As a small instability developed, an adapt led to increased resolution in the region of the instability, which stabilised the problem, as stated in the paper.

4. *Page 3237, line 16-20: Although direct convergence of these quantities is not expected due to the chaos of the underlying system, one may still anticipate that statistical convergence occurs. That is, over an ensemble of simulations one would expect that the mean head speed, etc. would be convergent with resolution. Can the authors comment briefly?*

This is an interesting discussion point. The turbulence is acutely sensitive to small perturbations in the initial conditions, parameters, and computational grid, and hence may produce quite a different result for any small perturbation of these. Therefore, it may be possible to obtain an ensemble average by making small perturbations to an initial condition, or very small changes to the mesh, around each of the mesh resolutions of interest. The authors agree that this has not been considered. The authors are not certain how many perturbations would be required to create a good ensemble average, how large these perturbations should be, and which parameters should be perturbed. This could turn in to an extremely complex, and interesting, but very expensive analysis.

The paper states that 'due to the turbulent nature of the flow, which is very sensitive to small changes in the mesh, it is very hard to show convergence of [many] quantities'. Based upon this discussion, the authors believe that the statements made in the paper referring to this are correct. We don't propose to make changes to the paper

GMDD

7, C770–C780, 2014

concerning this point, although believe that it is an interesting discussion topic.

5. *Page 3242, line 27: "Throughout the simulation the number of processor cores that were used was varied between 36 and 512 to keep the number of elements per core in the region of 20 000." Why? Does this have the effect of normalizing the wall-clock time of the simulation?*

The parallel efficiency of Fluidity increases as the number of elements per core increases. However, this relationship is not linear. It is generally accepted that the parallel efficiency of the program is good when there are more than $\approx 10^4$ elements per core.

Conversely, it is beneficial to have fewer elements per core as the simulation runs faster and we obtain results more quickly.

We do not want to run a very inefficient model and waste computer resource, but we do want the simulation to complete as fast as possible. The number of elements varies by approximately an order of magnitude over the course of the simulation. The number of cores that the simulation is run on is varied over a similar magnitude simply to optimise the computer usage.

Referring to the wall-clock time. All processing times given in the paper are processor hours e.g. a simulation running on 2 cores for 1 hour $= 2$processor hours.

6. *Section 6: I suggest the authors include an image depicting a snapshot of the adapted mesh near the gravity current head at an intermediate simulation time. Such an image would provide a better visualization of the effect of refinement on the mesh.*

The authors agree that this addition would improve the manuscript. We suggest including an extra paragraph in the 'benefits of adaptivity' section that describes Fig 1. (attached)

GMDD

7, C770–C780, 2014

Figure 1 shows the adapted mesh over a subdomain in the region of the current head at two times, $t = 3.5$ and $t = 4.0$. The images are generated from the three-dimensional simulation, and are taken from a y-normal plane at $y = 0$. The cut plane is chosen to be at the edge of the domain as a good two-dimensional representation of the mesh can be obtained at boundaries where all element surfaces are parallel to one another. These images demonstrate how the mesh adapts to the concentration field c, and velocity field, $\vec{u}$. The images also show how the mesh changes over a short period of the simulation. The change between $t = 3.5$ and $t = 4.0$ is dramatic. Very few, if any elements, within this view are consistent. The images clearly display how anistropic elements are generated along the density interface and the boundary layer where the curvature of the solution is highly anisotropic.

7. *Page 3247, line 24: Can the authors provide a brief discussion on the type of adaptive mesh refinement chosen for this model and how it compares to other, perhaps computationally cheaper techniques, such as a octree-based refinement or block-adaptive refinement?*

A mesh optimisation algorithm (Pain et al., 2001) is used in Fluidty for adapting the mesh. This algorithm offers the most flexibility in producing an optimised mesh, allowing for node movement as well as node insertion or deletion and edge/face swapping. Many dynamic features of the gravity current simulated are anisotropic e.g. the boundary layer and the density interface. The mesh optimisation algorithm used can easily provide a mesh with anisotropic elements such that these features are resolved efficiently. Therefore, this method of adaptivity is likely to produce a well optimised mesh for this application.

However, the adapt process takes a significant portion of the simulation time, hence it

GMDD
7, C770–C780, 2014

is a useful to consider the use of other, cheaper techniques. So long as the problem of hanging nodes can be addressed a hierarchical approach could be used, and may prove a suitable alternative. We propose adding a sentence in the middle of paragraph 3244:18-3245:2 to highlight this

> Adaptivity does come at a cost. The mean time required for a parallel adapt operation throughout this simulation, including mesh adaptation, data-transfer, mesh partitioning and data-migration, was 110s. This can be compared to a mean time required to compute a time step in parallel of 67s. Therefore, when adapting every 5 time steps, approximately $1/4$ of the simulation time is spent adapting the mesh, or the run time is increased by 33% compared to a fixed mesh simulation using the same number of elements. When adapting every 2 time steps approximately $1/2$ of the simulation is spent in the adapt stage. **The mesh optimisation algorithm used provides the most flexibility for mesh refinement, and hence will produce a highly optimised mesh, but it is potentially more expensive than other adaptivity algorithms. A high percentage of the total simulation time is spent in the adapt phase and hence it may be worth considering cheaper alternatives based upon hierarchical refinement for future models.** Regardless of this, the benefits of reducing the number of elements by two orders of magnitude far outweigh the cost of adaptivity. The total computing time was approximately $100,000$ hours of processor time. Over 500 cores this equates to just under a week of run time. Assuming a linear increase in run time with number of elements, a fixed mesh simulation would have taken at least an order of magnitude longer and would have been nearly impossible to post-process.

GMDD
7, C770–C780, 2014

We hope that we have answered your questions satisfactorily. Thank you again for a thorough and thought provoking review.

References

Espath, L., Pinto, L., Laizet, S., and Silvestrini, J.: Two- and Three-Dimensional Direct Numerical Simulation of Particle-Laden Gravity Currents, Computers & Geosciences, 63, 9–16, doi:http://dx.doi.org/10.1016/j.cageo.2013.10.006, 2013.

Farrell, P., Piggott, M., Pain, C., Gorman, G., and Wilson, C.: Conservative interpolation between unstructured meshes via supermesh construction, Computer Methods in Applied Mechanics and Engineering, 198, 2632–2642, 2009.

Georgoulas, A., Angelidis, P., Panagiotidis, T., and Kotsovinos, N.: 3D numerical modelling of turbidity currents, Environmental fluid mechanics, 10, 603–635, doi:10.1007/s10652-010-9182-z, 2010.

Huang, H., Imran, J., and Pirmez, C.: Numerical modeling of poorly sorted depositional turbidity currents, Journal of Geophysical Research, 112, 1–15, doi:10.1029/2006JC003778, 2007.

Imperial College London AMCG: Fluidity manual v4.1.11, http://dx.doi.org/10.6084/m9.figshare.995912, 2014.

Necker, F., Hartel, C., Kleiser, L., and Meiburg, E.: High-resolution simulations of particle-driven gravity currents, International Journal of Multiphase Flow, 28, 279 – 300, doi:10.1016/S0301-9322(01)00065-9, http://www.sciencedirect.com/science/article/pii/S0301932201000659, 2002.

Pain, C., Umpleby, A., De Oliveira, C., and Goddard, A.: Tetrahedral mesh optimisation and adaptivity for steady-state and transient finite element calculations, Computer Methods in Applied Mechanics and Engineering, 190, 3771–3796, 2001.

Sequeiros, O., Cantelli, A., Viparelli, E., White, J., Garc?a, M., and G., P.: Modeling turbidity currents with nonuniform sediment and reverse buoyancy, Water Resources Research, 45, 1–28, doi:10.1029/2008WR007422, 2009.

Interactive comment on Geosci. Model Dev. Discuss., 7, 3219, 2014.

CC BY

GMDD
7, C770–C780, 2014

Interactive Comment

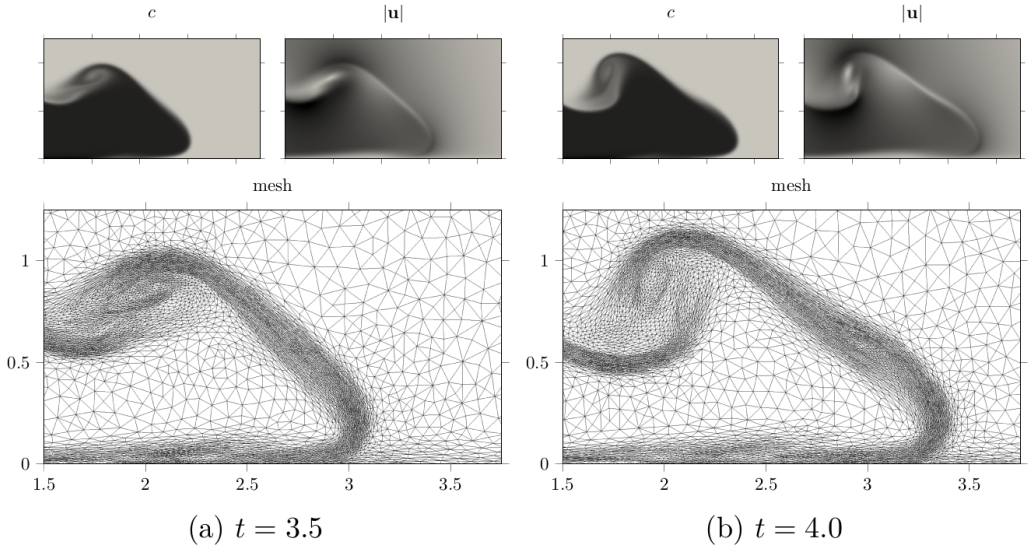

Fig. 1. Images showing concentration, c, velocity magnitude, |u|, and the adapted mesh at t=3.5 (a) and t=4.0 (b) over the subdomain, 3.5<x<3.75, z<1.25 on a y-normal plane at y=0.

Full Screen / Esc
Printer-friendly Version
Interactive Discussion
Discussion Paper