



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: 2 June 2014

The authors thank the referee for providing a thorough and thought provoking review, and 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.

Full Screen / Esc

Printer-friendly Version

Interactive Discussion

Discussion Paper



1 Major comments

3221-13: Please be accurate on giving references. Which of the cited papers refers to hundred km/hr turbidity current, Heezen and Ewing or Talling et al? The latter is mostly an analysis of turbidites, so I reckon it has to be the former refereeing to the Grand banks event?

Agreed. This reference is not totally accurate. Combined with a comment received from another reviewer concerning this sentence, we have modified the sentence to read as follows

A single submarine particle-laden density currents can involve $100km^3$ of sediment (Talling et al., 2007). That is approximately ten times the sediment flux into the ocean from all of the Earth's rivers combined (Talling et al., 2007). They can travel for hundreds of kilometres over the sea bed at speeds of tens of metres per second (Heezen and Ewing, 1952).

3222-16: Define Grashof number, explain physical meaning, and how it relates to Reynolds number. Gr number is used in lock-exchange problems, but not very often in other flow environments, where Re is much more popular.

This is a good suggestion that will make the paper more easily readable by a wider audience. We propose augmenting the sentence in question so that it reads

The scale of lock-exchange flows is typically described using the Grashof number. The Grashof number approximates the ratio of buoyant to viscous forces. This is equivalent to the square of the Reynolds number of a flow where the buoyancy velocity is used as the characteristic velocity. DNS

C706

Full Screen / Esc

Printer-friendly Version

Interactive Discussion

Discussion Paper



modelling of particle-laden density currents has been achieved in three dimensions at moderate Grashof numbers of

3222-21: Is this concentration volumetric or by weight? Define here.

It is volumetric. We propose adding some information to clarify this by replacing

... modelling of low particle concentration ...

with

... modelling of low volumetric particle concentration ...

3223-11: Is a Gr number of 5×10^6 turbulent enough? A rough square root conversion yields $Re=2236$, which can be turbulent but also transitional between laminar and turbulent depending on the circumstances and the problem. This issue deserves an analysis.

Necker et al. (2002) found the flow to be fully turbulent at this Grashof number, as is the case in this work, so $Gr = 5 \times 10^6$ is high enough to obtain a fully turbulent flow. Specifically answering your question 'is it high enough?' depends upon exactly what the aim of the work is. Obviously it would be nice to increase the Grashof number if possible, and this will be the aim of future work.

We follow Necker et al. (2002) in using $Gr = 5 \times 10^6$ for the purposes of comparing computational results. It is a highly cited paper in the field, and is a simulation set up that has been used for validation and comparison by both Nasr-Azadani and Meiburg (2011) and Espath et al. (2013) for similar purposes.

We agree that the issue deserves an analysis and there are other papers that look at simulations over a range of Gr numbers (Espath et al., 2013; Cantero et al., 2007, 2008) and analyse the difference in results obtained.

The aim of this paper is not to validate the use of a particular Grashof number. However, we propose modifying the last sentence of the paragraph in question to read as below

Even at these moderate Grashof numbers a fully turbulent flow is obtained (Necker et al., 2002; Espath et al., 2013), and very useful insights have been obtained from simulations of gravity currents within this range.

3223-15: Be more specific about what 'large domain'. It is realistic to expect using Fluidity in domains larger than the experiments replicated in this manuscript. But probably field scale modeling-order of magnitude of dozens of kilometers, is still out of reach. Please comment.

We agree in part that this is not adequately explained. It is important to distinguish between higher Grashof numbers and larger domains. This comment is meant to imply larger domains with similar Grashof numbers. This is explained more fully in the conclusion. We suggest modifying the end of the introduction to read

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.

3225-8: Is 'displacement effect' equivalent to 'added mass effects'? Or is it something else?

[Full Screen / Esc](#)[Printer-friendly Version](#)[Interactive Discussion](#)[Discussion Paper](#)

This is not the same as 'added mass'. This is simply the displacement of fluid by particles, which will increase the fluid volume and invalidate the incompressibility assumption. We propose modifying this section of text to read

The fluid is assumed to be incompressible. Due to the very low volumetric concentrations, the displacement of fluid by the suspended particles can be ignored (Necker et al., 2002). Therefore the velocity field is considered to be divergence-free.

3227-equation 8-line 11: Is η the bed elevation rather than the volume? Otherwise the units in equation 8 do not work well. Also, how is the porosity accounted in equation 8? In this equation - which is also known as Exner equation, porosity is usually accounted on the Left Hand Side modifying $d\eta/dt$.

You are correct. η is the bed elevation. Thank you for spotting this mistake. We do not consider porosity. Porosity is not usually considered when calculating deposit depth from models of this type. See Equation (21) in Necker et al. (2002), Equation (14) in Espath et al. (2013). The comparative experimental results of De Rooij and Dalziel (2009) are given in g/cm^3 such that an estimation of porosity is not necessary. It is also worth noting that the maximum deposit in the experiments of De Rooij and Dalziel (2009) is $0.018g/cm^3$ which gives a depth of $\approx 68\mu m$. This is only just equal to a single layer of particles and hence porosity does not have any impact on the result.

However, this is an interesting comment. We have agreed that this omission of porosity should be noted in the manuscript. We propose to add a comment at the end of the paragraph 3226:17

... boundary unit normal vector. As in the work by Necker et al. (2002) and Espath et al. (2013), no adjustment for porosity of the deposit is included.

Full Screen / Esc

Printer-friendly Version

Interactive Discussion

Discussion Paper



3229-15: I don't think the authors have explained what "discontinuous" means in the context of the Galerkin FEM scheme. A brief explanation of highlighting the differences of Continuous vs Discontinues Galerkin scheme no longer than a short paragraph will help.

A useful point. We propose to augment the paragraph in question as

... A linear Discontinuous Galerkin (DG) scheme is used for the discretisation of both the velocity and sediment concentration fields.

A DG discretisation does not enforce continuity across element boundaries. A field that is discretised on a DG function space may therefore have multiple values at element boundaries. It will also have more degrees of freedom than a CG function of the same order as elements do not share nodes. DG methods are a good choice of discretisation for advection-dominated problems as they produce stable discretisations without the need for stabilisation strategies such as streamline-upwinding (Peraire and Persson, 2008)...

3229-16: are velocity and sediment concentration discretized in different fields?

We use the term 'field' to refer to a scalar or vector field within the domain. As such, velocity and sediment concentration are different fields. They do, however, both use the same discretisation.

3231-9:15: A flow chart with pictures of grid examples at each stage is need to follow the complex adaptivity system described in 2.3 with metric formation, new mesh generation and data transfer.

We agree that a flowchart may help to illustrate the adapt algorithm and propose including the flow chart shown in the attached figure (Fig 1). This will be referenced in

C710

Full Screen / Esc

Printer-friendly Version

Interactive Discussion

Discussion Paper



the introduction text to the adaptivity section.

3235-5: Why are velocity free-slip b.c. used at the wall sides? Is it for the sake of comparison with other papers or to minimize the number of elements?

The reason behind using free slip boundary conditions for the side walls is to keep the simulation set up comparable with the other modellers simulations. We feel that this question is already answered at the end of the paragraph in question.

3237-21: 'However, one important quantity does show convergence.' Which one? Say it here.

The quantity that shows convergence is ϵ_d . We propose a modification of this sentence to

However, one important quantity does show convergence. This is the energy lost due to discretisation, and data transfer errors.

NOTE: there is a typographical error at 3238:2: ϵ_E should be ϵ_d

3237:24:28: "The combination of upwind flux terms and slope limiting in the discretisation dissipates energy at scales that the mesh cannot resolve. Additionally, adapting the mesh requires a data transfer operation which produces errors in conservation of energy, although these are minimal compared to the numerical dissipation from an under-resolved mesh." This paragraph is a bit dark. Are you saying that the errors in energy conservation due to data transfer from mesh to mesh are negligible compared to errors due to the mesh not being small enough?

There will be some error due to data transfer using 'Galerkin projection', but these errors are very small. An under-resolved mesh has the capability to introduce huge

Full Screen / Esc

Printer-friendly Version

Interactive Discussion

Discussion Paper



errors comparatively. Obviously the goal is to get both sources of error to be small enough such that they are negligible.

We propose re-wording the paragraph to make it clearer, as follows

... The combination of upwind flux terms and slope limiting in the discretisation dissipates energy at scales that the mesh cannot resolve. Additionally, adapting the mesh requires a data transfer operation which will introduce some relatively small errors. By computing the energy budget ...

3244-3: Why this high resolution clustered on the left hand wall and not on the right hand wall? Shouldn't it be more or less symmetric?

An interesting point. Referring to Fig. 7. At $t = 8$ the current is very close to the left-hand wall. The fluid in front of the current has a very wide area to recirculate, the eddy is much larger, and hence the curvature in velocity is much less. At $t = 20$ the head of the current is beyond half-way along the domain but the body still extends back to $x = 2$. The area available for recirculation behind the current is still significantly smaller than the area in front of the current.

Note that the right-hand wall is not actually shown in the Figure 7. The total domain length in x is 19. We propose to add this detail as a note in the caption to make it clear, as follows

Heatmap indicating the size of the elements in x , y and z across a plane at $y = 1.0$ for a subset of the domain ($-1.0 < x < 12$) and times $t = 8$ (left) and $t = 20$ (right). Note that the domain extends to $x = 18$. The region $12 < x < 18$ had no significant regions with element sizes smaller than 0.1

We propose modifying the text around 3244-3 to read

C712

Full Screen / Esc

Printer-friendly Version

Interactive Discussion

Discussion Paper



There is also high resolution around vortices and at the left hand wall where there is a strong recirculation of the overlying fluid. The increase in resolution at the left hand wall is present at both $t = 8$ and at $t = 20$. The same increase in resolution is not present at the right hand wall at either $t = 8$ or $t = 20$ (not shown in Figure 7). This is because the fluid has more space to recirculate in front of the current than behind it in both cases such that the curvature of the velocity field, and thus the mesh resolution, is less.

3246-3:4: “The addition of erosion of the bottom surface in this simulation will increase the concentration of the head of the flow and may lead to a faster head speed.” If this happens it is only an initial transient effect due to the inertia of the dense flow collapsing as the lock opens. With zero bed slope no flow can sustainably pick up sediment and self-accelerate (Sequeiros et al. 2009). The fact that deposition steadily increases from the very beginning until beyond $t=10$ (Figure 10) hints that self-acceleration is not occurring.

We agree with your comment and will remove this sentence.

3246-16:24: This paragraph is misleading. A better agreement with previous studies does not imply that the outcome is real. It just points to the numerical models doing the same thing. Artificially vanishing entrainment does not make it happen in the real world.

This paragraph is not intended to imply that anything happens in the real world. We are simply trying to show that the deposition rate results from the Fluidity model match well with other models. This requires removing the effect of the erosion rate. Hopefully the following paragraph will work better at explaining this

Noting that the vertical fluid velocities are small near the bed due to the

no slip boundary condition, and that eroded sediment will be settling, the majority of eroded sediment will almost immediately be deposited, and will never be fully entrained back into the flow. This will lead to an increased deposition rate compared to a simulation without erosion. By making the assumption that all eroded sediment is immediately deposited, a modified deposit rate can be calculated for the Fluidity simulation with the effect of erosion removed. As shown in Figure 10, this modified deposition rate shows much better agreement with the results of both Necker et al. (2002) and Espath et al. (2014) leading to the conclusion that it is the inclusion of erosion in the simulation that led to the higher deposition rate.

3247-1:4: This comment on Figure 11 is also misleading. The other models do not have erosion capabilities (as stated in 3246-13:15) and their outcomes in Figure 11 are similar to Fluidity's. It seems to me that the three models simply cannot match the experimental results. This is regardless of the algorithm for erosion included or not. Ergo the problems seem to be something intrinsic in the models themselves beyond them having capacity to model some degree of erosion with empirical equations or not.

We can appreciate how there is some confusion about what is being said here. We do not aim to say anything about a match with De Rooij and Dalziel (2009) other than that the peak deposit is in approximately the right place. The comments mainly focus on comparison with the other computational models. We agree that this analysis could be improved and made clearer and propose the following modification

An important diagnostic for applications is the final deposit profile from a particle-laden density current. Figure 11 shows the span-wise averaged deposit profile from the three-dimensional Fluidity simulation compared against those of previous modellers, and also from the experiments

of De Rooij and Dalziel (2009). A good match is observed in the peak deposit height of $\eta \approx 0.12$ at $x \approx 4$ between all of the models and the experimental results.

There is a notable variation in deposit depths near the lock-gate. All models show a smaller deposit depth in this region when compared to the experimental results. The reason for this is unclear and explanations can only be speculative. One potential cause may be that the sediment in the experimental set up had already begun to settle before the lock-gate was released. This may also help to explain the slightly shorter run-out distance resulting from a reduced initial potential energy. Alternatively, there may be processes occurring in the laboratory that are not accurately captured by the computational models.

The results from Fluidty are further from the measured results than the other models in this region. The inclusion of an erosion algorithm is the likely cause of this. The experimental measurements show larger deposits than all of the models upstream, and smaller deposits downstream. Erosional processes will predominately decrease upstream deposits and increase downstream deposits, and hence would increase this discrepancy if applied to any of the models. In addition to this, the erosion algorithm is not configured correctly to match the De Rooij and Dalziel (2009) experiment. $R_p \approx 1$ for the De Rooij and Dalziel (2009) experiment, in comparison to $R_p \approx 20$ in the Fluidity simulation. This will result in significantly more erosion in the simulation than is likely to have occurred in the experiment.

[Full Screen / Esc](#)[Printer-friendly Version](#)[Interactive Discussion](#)[Discussion Paper](#)

2 Minor comments

We believe that 'scale' is the correct word to use at 3220-17 and so do not propose to change it here. Apart from this, We agree with all of the minor comments and will make the changes requested. We will also add an acknowledgement to the reviewers for helping to improve the manuscript.

Thank you again for a thorough and thought provoking review.

References

- Boman, E. G., Çatalyürek, Ü. V., Chevalier, C., and Devine, K. D.: The Zoltan and Isorropia parallel toolkits for combinatorial scientific computing: Partitioning, ordering and coloring, *Scientific Programming*, 20, 129–150, 2012.
- Cantero, M., Lee, J., Balachandar, S., and Garcia, M.: On the front velocity of gravity currents, *Journal of Fluid Mechanics*, 586, 1–39, doi:10.1017/S0022112007005769, 2007.
- Cantero, M., Balachandar, S., Garcia, M., and Bock, D.: Turbulent structures in planar gravity currents and their influence on the flow dynamics, *Journal of Geophysical Research (Oceans)*, 113, C08 018, doi:10.1029/2007JC004645, 2008.
- Chen, L., Sun, P., and Xu, J.: Optimal anisotropic meshes for minimizing interpolation errors in L^p -norm, *Mathematics of Computation*, 76, 179, 2007.
- De Rooij, F. and Dalziel, S. B.: Time- and Space-Resolved Measurements of Deposition under Turbidity Currents, pp. 207–215, Blackwell Publishing Ltd., doi:10.1002/9781444304275.ch15, 2009.
- 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. and Maddison, J.: Conservative interpolation between volume meshes by local Galerkin projection, *Computer Methods in Applied Mechanics and Engineering*, 200, 89–100, 2011.

Full Screen / Esc

Printer-friendly Version

Interactive Discussion

Discussion Paper

- Heezen, B. C. and Ewing, W. M.: Turbidity currents and submarine slumps, and the 1929 Grand Banks [Newfoundland] earthquake, *American Journal of Science*, 250, 849–873, 1952.
- Hiester, H., Piddott, M., and Allison, P.: The impact of mesh adaptivity on the gravity current front speed in a two-dimensional lock-exchange, *Ocean Modelling*, 38, 1–21, doi:10.1016/j.ocemod.2011.01.003, 2011.
- Nasr-Azadani, M. M. and Meiburg, E.: TURBINS: An immersed boundary, Navier–Stokes code for the simulation of gravity and turbidity currents interacting with complex topographies, *Computers & Fluids*, 45, 14–28, 2011.
- 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.
- Peraire, J. and Persson, P.: The compact discontinuous Galerkin (CDG) method for elliptic problems, *SIAM Journal on Scientific Computing*, 30, 1806–1824, 2008.
- Talling, P., Wynn, R., Masson, D., Frenz, M., Cronin, B., Schiebel, R., Akhmetzhanov, A., Dallmeier-Tiessen, S., Benetti, S., Weaver, P., Georgiopoulou, A., Zuhlsdorff, C., and Amy, L.: Onset of submarine debris flow deposition far from original giant landslide, *Nature*, 450, 541–544, doi:10.1038/nature06313, 2007.

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

GMDD

7, C705–C718, 2014

Interactive
Comment

Full Screen / Esc

Printer-friendly Version

Interactive Discussion

Discussion Paper



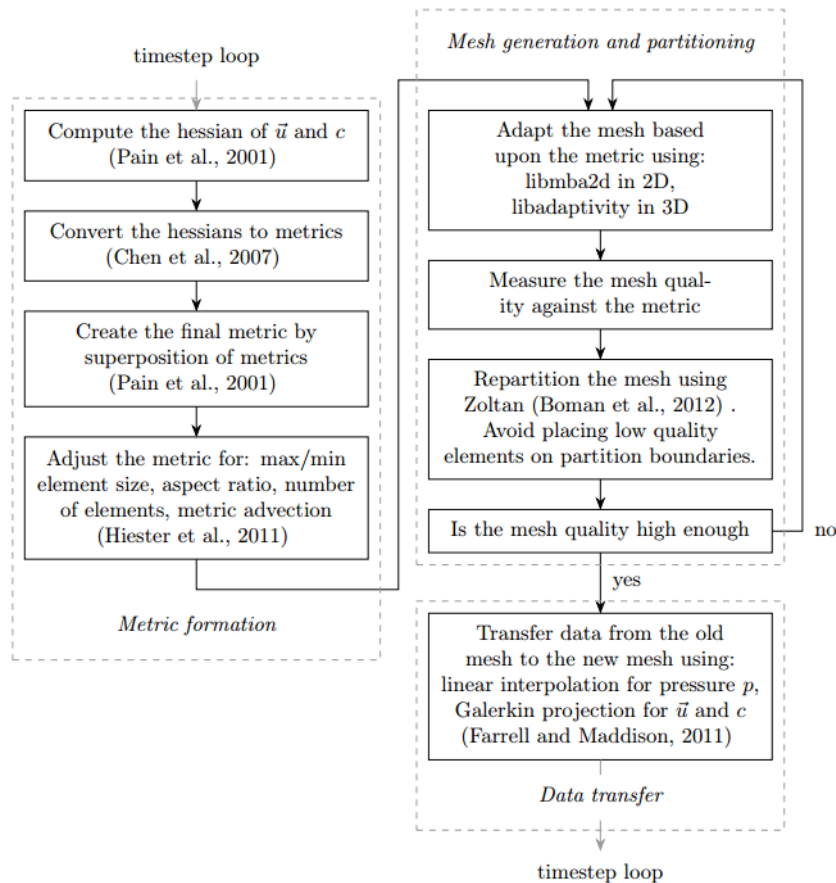


Fig. 1. A description of the high-level algorithm involved in adapting the mesh. This algorithm is invoked repeatedly throughout the simulation at a fixed interval specified as a number of timesteps.