The Cryosphere Discuss., 4, C1353–C1363, 2010
www.the-cryosphere-discuss.net/4/C1353/2010/
© Author(s) 2010. This work is distributed under
the Creative Commons Attribute 3.0 License.

The Cryosphere Discussions

Interactive comment on "Parameterization for subgrid-scale motion of ice-shelf calving-fronts" *by* T. Albrecht et al.

T. Albrecht et al.

torsten.albrecht@pik-potsdam.de

Received and published: 13 December 2010

Response to Referee: Dan Goldberg, (dgoldberg@cims.nyu.edu)
Received and published: 16 November 2010

General comments:
This paper presents the numerical handling of an advancing ice shelf front, and is presented as a companion to several other papers dealing with modifications that the authors have made to the model PISM. This paper does not deal with a specific calving parameterization, which can be seen as the "physical" side of the calving problem, but rather how to deal with the "technical" side: given a calving parameterization, there

is a velocity associated with the ice shelf front (either positive or negative), and if the ice momentum and mass equations are solved on a rectangular grid, then they must account for this velocity and must reproduce analytical solutions wherever available. What I like about this paper is that the authors bring attention to this "technical" side of calving; in the literature on calving studies I have seen not much mention is made of it, and yet dealing with it is a prerequisite in studying and investigating calving parameterizations. For example, the problem of an "overly diffusive" front is one that I am not sure all ice shelf modelers are aware of, but certainly should be. The fact that they chose a very simple parameterization to work with might not be a problem.
That being said, there are some issues on which the manuscript is unclear and/or confusing. The treatment is described in great detail for the 1D (flowline) model, but the 2D treatment is given almost as an afterthought. This is unfortunate, since the 2D treatment is, or should be, the main contribution of this manuscript. Developing the 1D parameterization oneself would not be that difficult (and, by itself, not worth writing a paper about). Detailed comments follow, and typographical/grammatical errors are at the end.

Section 1 Comments:
paragraph beginning page 1498, line 18: in presenting this issue, you have already assumed a numerical discretization, no matter how common. I have no idea how a 10th-order polynomial finite-element solution would behave in such a situation. You should present it as an issue with the chosen discretization.

We discuss the numerical scheme later in Sect. 2 of this manuscript and in more detail in the companion model description paper (Winkelmann *et al.* (2010), Eq. 2.5.1+2). We agree and added some clarifying formulations to discuss the diffusion problem as an issue of the used finite difference scheme. The diffusion effect and the related problem of applying the stress boundary condition might not be a problem for finite-element.

Section 2 Comments:
Eqn 3: This looks like a CFL condition that would apply to a 1D model. Is the same applied to the 2D version of the advection equation? Or is there a numerical factor?

We added the 2D CFL criterion to the manuscript, which is basically the sum of the 1D CFL condition for each axis with an additional small term to avoid division by zero (as in the PISM base code):

$$\Delta t_{adapt} = \min_{i,j} \left(\frac{|u_{i,j}|}{\Delta x} + \frac{|v_{i,j}|}{\Delta y} + \frac{\varepsilon}{\Delta x + \Delta y} \right)^{-1}. \qquad (1)$$

Section 3 Comments:
Overall: There is subscripting in this section to denote which grid cells are being discussed. But there is no mention of ***when*** *things are updated, initialized, etc. What is the order of operations in your algorithm? e.g. do you calculate R for partially-filled cells at the end of a timestep, once thicknesses are updated (and therefore using the new value of H_c to find H_r)? My guess would be that over a given timestep, you (0) have the old value of thickness, R, and H_r (H^n, R^n, H_r^n), and you first (1) calculate velocities (u^n), then (2) advect thickness to get H^{n+1}, (3) determine which cells are now partially filled based on H_r^n, and finally (4) calculate R^{n+1} for the partially- filled cells, using new values of H_r determined from H^{n+1}. (superscripts denote time level.) This seems to be the most direct order of operations, but then again it may be problematic (see below). And anyway I should not have to guess.*

We comprehend that the manuscript may cause difficulties for a reader to understand the exact chronological algorithm of the procedure. Since our purpose is to give a general idea of the parameterization and its generic behavior we choose a different structure of the description and avoided time indices to keep the clear view.

In the manuscript we added temporal terms such as “old”, “new” and “previous time step” instead to clarify the order.
And the referee’s guess is almost correct: (1) Velocities are calculated only the basis of the “old” ice thickness distribution H^n with applied “old” boundary conditions (0). The “new” time step is fixed or given by the generalized CFL-condition (Eq. 1) to calculate the mass fluxes, also the ice flux through the “old” boundary (determined by a special mask), which gives a “new” ice volume V^{n+1} there. This operation results in a “new” ice thickness H^{n+1} throughout the ice shelf (2) but we calculate the H_r^{n+1} as average over the “old” H^n for the partially filled grid cells (3). Hence we can calculate a new $R^{n+1} = V^{n+1}/(a\,H_r^{n+1})$ (4). If $R^{n+1} < 1$, the “new” local boundary position equals the old one (for applying boundary conditions and calculating fluxes in the next time step) and the ice front position can be thought to be located somewhere within the partially filled grid cell depending on R^{n+1}. If it exceeds 1 the residual ice mass can be redistributed to neighboring partially filled grid cells and H_r^{n+1} becomes H^{n+1}, which changes the local boundary position for the next velocity calculation by a grid cell length. This all happens on the time level n+1.

p1503, line 22: where do the values of V_{i+1} come from? That is, you clearly have a velocity at each filled/partially-filled interface which is negative and larger than the ice velocity – but which thickness is used to find volume flux? H_r? This should be mentioned as it is part of the parameterization.

Right, this will be mentioned in the revised manuscript. We define a calving rate so far as a “negative” velocity magnitude, which is though to be applied directly at the sub-grid ice front somewhere within the partially filled grid cell, so H_r or $H_{r,red}$ is used in this case to calculate the volume loss. This means, we compare volumes/masses within the partially filled grid cell here instead of velocities defined at the boundary.

p1503, paragraph beginning line 26: This is an instance where insufficient detail *is given on the 2D treatment. The issues I can think of: (1) Is it not possible that this could result in H_r being thicker than one of the adjacent cells? Is this not an issue when the cell becomes filled and is now thicker than an upstream cell? (2) With the CFL condition as in eqn (3), this could result in, say, an empty cell ($R = 0$) becoming overfilled in a single step, could it not? And a similar issue for retreat, if a full cell is drained by more than one partial cell? (3) For retreat: does each interface have an associated volume flux, as in p1504,line 22? If a partial cell is completely drained, how is the excess (negative) volume partitioned to multiple adjacent full cells? (4) If there are 3 adjacent cells, H_c is averaged over all 3 to get H_r?*

We totally agree, that 2D is the more interesting part and we are aware of such cases the referee is concerned about. They occur and we account for this as explained below and we adjoined some more explanations to the manuscript. But we don't think that it is necessary to evaluate all possible issues in the framework of this publication. The related physical error is small in simulations we ran, probably due to the smoothness of velocity and hence of ice thickness distributions in ice shelves in the frontal area.

(1) This is definitely possible, but it is not a big issue, when the time steps are sufficient small to allow for some kind of relaxation of the new terminal ice thickness (balance of influx and outflux) before the next row of partially filled grid cells becomes full. If it occurs only for single grid cells the effect on the velocity distribution is negligible.
However, in the case of adaptive time steps two things can happen in areas along the front with maximum terminal velocities. If H_r is much thicker than the upstream cell and this partially filled grid cell becomes full, this anomaly can be advected forward with the front, which is similar to the described wiggle phenomenon. The $H_{r,red}$-formulation reduces this effect.
(2) If H_r is much smaller, it may occur that an empty grid cell becomes more than

full within a time step. We account for that excess mass by redistributing this mass to the additional still partially-filled or empty neighbors (if using variant 2), before the next time step starts. CFL just limits the amount of excess mass that needs to be redistributed (in an unphysical manner), but if there is still unaccounted excess mass the redistribution procedure is done one more time.
(3) As the calving rate we use has no direction yet (just the magnitude) the excess mass loss is equally redistributed among neighboring full grid cells. It does not depend on the directed flux through each interface.
(4) This is correct. And we wanted to keep it simple. A more advanced version is to calculate a flux-weighted average for H_r, which assures that an empty grid cell, which becomes filled in one time step produces no excess mass (and H_r should not be thicker than the upstream grid cell). But since we have in general partially filled grid cells, that may produce excess mass anyway, the benefit is not so prominent.

Eqn(8): I have several comments/questions about this $H_{r,red}$ formulation - (1) If a cell is filled over the course of a timestep, is its thickness then set equal to H_r (or $H_{r,red}$, whichever is being used)? Is this thickness then the H_c that is used to find H_r, and R, for the new partially-filled cell? If not, please explain the process in more detail. But if so, I can see a way that a completely ice-free cell can become completely filled (with excess mass that needs to go into an additional cell). CFL would not allow the cell to become completely filled with $H_r = H_c$, but if H_r is then adjusted according to eqn (8), R could increase and potentially become larger than 1. Perhaps this is not possible, but a better explanation of the algorithm (re: my comment about order of operations) would make this clear. (2) It is not clear how this would work in 2D, since each neighboring cell would give a different $H_{r,red}$. (3) In a polythermal version, would you adjust the parameter C?

(1) The $H_{r,red}$ formulation is in fact only used for the first row of adjacent partially filled grid cells (if variant 2). The thickness of the grid cell, that becomes full

(or even overfull) is set to $H_{r,red}$, which is generally closer to the "balanced" ice thickness (which gives a larger R). If there is additional excess mass unaccounted we can calculate new H_r (not $H_{r,red}$) based on the new terminal ice thicknesses (the former $H_{r,red}$). This is done until all excess mass is redistributed before the new time step starts. Another simple way would be, to reduce the adaptive time step by a flux-dependent factor.

(2) H_r is the average over the ice thickness of the adjacent full grid cells. This value will be reduced by a subtrahend that depends on the fifth power of H_r itself (We replaced in Eq. 8 H_c by H_r as the general 2D case). Thus, there is only one $H_{r,red}$ for each partially filled grid cell.

(3) This can be done of course, but the here used estimate seems to be sufficient for a broad range of boundary conditions Q_0, C and n, even though the ice shelf is actually buttressed or polythermal. It was introduced just to reduce the occurrence of a propagating ice wall in the adaptive time step case and is not more than a guess.

Section 4 Comments:

In this section, I have no problem with the material up to eq. 12. However, I find parts of the descriptions of the experiments unclear. I find it easier to organize my comments by experiment, not by page/line number. I hope that is alright.

The Experiment corresponding to Fig. 4: You say (p1507, line8): "Its [the ice-shelf extension scheme used in PISM] effect on the ice-shelf propagation is shown in comparison to a model result with applied CFBC in the flowline case (Fig. 4)." And so I expect that the "no CFBC" case in Fig 4 implements the shelf extension scheme described in the introduction. But p1507, line 11 says that the shelf extension scheme is not used. So is the shelf extension scheme applied in any of the experiments? And if not, what is used in the case where the CFBC is not used? The ice shelf needs a condition at each boundary. The upstream boundary condition is one of velocity; if the CFBC is implemented then there is a stress condition at the front. If I understand

the shelf extension correctly, the stress condition is applied at the boundary of the computational domain (not the ice shelf). But it is still applied somewhere. If you don't use the CFBC or shelf extension scheme, what is the seaward boundary condition of the equation solved for velocity?

Correct, it is the shelf-extension scheme, which is compared with the case, where the CFBC is evaluated directly at the front (p1507, line 11 has been changed in the manuscript). In both cases the sub-grid parameterization of ice front motion was used. Otherwise the velocity field would be also affected by the broadening effect of the front. In both cases at the upper boundary a Dirichlet boundary condition is applied. The CFBC we use is a Neumann boundary condition directly applied at the local terminal boundary. In the PISM base code extension scheme, however, on the ice-free ocean, the product of viscosity and ice thickness is set to a fixed constant value and there should be a Neumann condition at the computational domain, but PISM base code just uses periodical boundary conditions (as far as I know). In our experiments, however, we use Dirichlet boundary condition there (u,v=0). We will mention that in the manuscript to avoid confusion.

Also, I don't understand the sentence beginning p1507, line 18: as written it does not make sense.

"on the ice-free ocean" was discarded in this phrase.

The Experiment corresponding to Fig. 5: p1508 l14: either do not say "(higher order terms in approximation)" or explain what you mean.

We have reformulated this in the manuscript. Meant is the truncation error of the Taylor approximated transport scheme, which is of the order $O(\Delta x)$, which get smaller for finer resolutions, and hence the numerical solution is closer to the analytical one.

The Experiment corresponding to Fig. 6: For the most part I thought this was clear, and the results shown for variant 2 are interesting. My question, again, is about "variant zero": what is the boundary condition on the velocity solve if neither CFBC or shelf extension are used?

We clarified this in the revised manuscript. In variant 0 the PISM base code shelf extension without CFBC is used and in contrast to the experiment in Fig. 4 no sub-grid parameterization of the ice front motion is applied. Thus, variant 0 is basically the default PISM result without our additional code modifications as presented in this paper and its companions, but with vanishing velocities at the boundary of the computational domain.

2D Experiments: As I mentioned before, these are more interesting and not much text is devoted to explaining them. Are these steady states? Are you again using the 250-m calving rule? Which "variant" are you using? How is the boundary condition at the grounding line decided upon? How far is this configuration from an initial condition? Can they not be compared against the actual ice shelf fronts? (when the ice shelves existed, of course...)

The 2D case is definitely the more interesting case, but the basic principles of the sub-grid parameterization of the ice front motion can be far easier explained and tested in 1D and the extension of the concepts to 2D is not too complicated. We actually ran simplified 2D experiments ("flow around a corner" and "advance and retreat in different angles with respect to the grid").
We did not want to go into the details of the realistic regional ice shelf models, which we show in Fig. 7 and 8 and into their boundary conditions, since this was planed for a further publication about a strain-rate based calving law, we used here. These

are actually steady states with this calving law applied and we used variant 1 of the parameterization, which in fact does not matter so much for the steady state. It was more thought to get a glimpse into how we use the parameterization in our model studies.

Typos and grammatical errors:
p1499 l4: drives
l10: does not agree
p1502 l15: generalize
p1503 l13: remove "also"
l14: replace "was" with "is"
p1507 l12: "shows"
l12: m/a
p1508 l2: three-step
l10: you already said this.
l20: not "stated". "seen" maybe?
p1509 l8: extension

Was changed in the manuscript.

References

Winkelmann, R., Martin, M. A., Haseloff, M., Albrecht, T., Bueler, E., Khroulev, C., & Levermann, A. 2010. The Potsdam Parallel Ice Sheet Model (PISM-PIK) - Part 1: Model description. *The Cryosphere Discussions*, **4**(3), 1277–1306.