Title: ASAM v2.7: A compressible atmospheric model with a Cartesian cut cell approach
Author(s): M. Jähn et al.
MS No.: gmd-2014-110
MS Type: Model Description Paper
Iteration: Revised Submission

In our revised manuscript, we marked all changes in red color for verifiability. Again, the authors like to thank the reviewers and the topical editor for their thoughtful comments and suggestions to improve the quality of the paper.

Point-by-point respsonses to referee comments

L. Bonaventura (Referee)

1)

a) **Referee comment:**
A general critical comment is that the paper seems to aim at the same time at testing a novel numerical technique AND documenting a model with full physics. Although the most common (and in my view best) practice is nowadays to discuss the backbone numerical core first, with more emphasis on the mathematical properties, and to apply it to more realistic problems once its pros and cons have been clearly assessed, the authors are perfectly justified in pursuing the more ambitious goal of achieving both aims at once. However, the paper in the present form is unbalanced, missing a lot of detail on the numerical part (as it will be discussed later). Given the premise of an insufficient description and validation of the fundamental numerical method, it does not make much sense to present realistic results with full physics. On the other hand, the appendices and the detailed description of the subgrid scale parameterizations are adequate for a full model presentation, but they are a bit off the point in a paper focused on a new numerical technique. The authors may consider splitting the paper in two and testing the adiabatic dynamics part first.

b) **Our answer:**
Paper design:
Parts of the used methods are already described in other papers like

Hinneburg, D., Knoth, O. (2005): Non-dissipative cloud transport in Eulerian grid models by the volume-of-fluid (VOF) method. Atmos. Environ., 39, 4321-4330.

Doyle, J. D., Gaberšek, S., Jiang, Q., Bernardet, L., Brown, J. M., Dörnbrack, A., Filaus, E., Grubišić, V., Kirshbaum, D. J., Knoth, O., Koch, S., Schmidli, J., Stiperski, I., Vosper, S., Zhong, S. (2011): An intercomparison of T-REX mountain wave simulations and implications for mesoscale predictability. Mon. Wea. Rev., 139, 2811-2831.

Jebens, S., Knoth, O., Weiner, R. (2011): Partially implicit peer methods for the compressible Euler equations. J. Comput. Phys., 230, 4955-4974.

In our paper, we would like to give a short overview about the actual numerical implementation without going too much into detail, e.g. how it has been done for the DALES model:

T. Heus, C. C. van Heerwaarden, H. J. J. Jonker, A. Pier Siebesma, S. Axelsen, K. van den Dries, O. Geoffroy, A. F. Moene, D. Pino, S. R. de Roode, and J. Vilà-Guerau de Arellano (2010): Formulation of the Dutch Atmospheric Large-Eddy Simulation (DALES) and overview of its applications. Geosci. Model Dev., 3, 415-444.

However, we would like to present the main flavor of the algorithms, where not all of them are mathematically profounded.

a) **Changes in the manuscript:**
Our original aim was present a monolithic atmospheric model (which has not been described as a whole in the literature so far) with an advanced time integration method and a cut cell approach in space, which overcomes the small cell problem and gives comparable results for test examples from the literature. Thus, the main intention of our paper is to give an overview of the main ingredients of the model implementation with a list of test examples. However, we agree that this approach might be a little bit too ambitious since there would be a lack of depth regarding the description of the methods.
We have restructured paper in the following way: The focus now lies on the description of the methods (Sections 2.2 and 2.3) plus suitable test examples (Section 4). We still decided to keep the 'model phyiscs' section so the reader can get a knowledge about the most important parameterizations that are implemented in the model. Furthermore, they are partly important for the last test case.
In addition, we reworked introduction with nearly the double amount of citations to better fit our model into the current literature.

2)

a) **Referee comment:**
This is apparently the first attempt to describe properly the proposed shaved cell approach in the published literature (at least, no previous reference is given by the authors). Therefore, a full and detailed account of all the aspects of the method should be given. In particular, it should be clearly explained of how 'special' cell configurations are handled. The authors instead simply mention in the caption of figure 1 that some possible configurations are 'excluded', and classify such configuration by the number of 'markers' without defining this term, that is only used in the captions of figure 1. The authors should clearly describe the mesh pre-processing approach they employ, define all the relevant quantities and also explain in detail how the staggered control volumes for the momentum variables are defined, which is not at all obvious on this kind of meshes.

b) **Our answer:**
Shaved cell method
For the spatial discretization only the six partial face areas and the partial cell volume and the grid sizes of the underlying Cartesian mesh are used. For a proper visualization we smooth the orography in such a way that the intersection of a grid cell and the orography can be described by a single possible nonplanar polygon. Or in other words, a cartesian cell is divided in at most two parts, a free part and a solid part.

c) **Changes in the manuscript**
As mentioned, we reworked Section 2.2. For a better overview, subsections are introduced in 2.2. Also, additional figures were added (Figs. 4+5) to make the approach more clear.

3)

a) **Referee comment:**
Several other important details concerning the numerical method are omitted. In particular, all the points below should be explicitly addressed in a revised version:
a) It is said that the method is a mixture of finite volume and finite difference approaches, but the model equations are in flux form and the only reference to finite difference approximation is the sentence 'The pressure gradient and the Buoyancy term are computed for all faces with standard difference and interpolation formulas with the grid sizes taken from the underlying Cartesian grid.' (at the end of section 2.2). However, if the 'shaved cell' structure is ignored when computing the pressure gradient, serious inaccuracies may result.It is up to the authors to prove that this is not the case, but they do not present any stringent accuracy assessment close to the lower boundary (see point 5). It is unclear why the pressure term is not included in the flux formulation, thus leading to a full finite volume formulation (for which however the issues discussed below in 3.b and 3.d would also be relevant). The authors should clarify this point and (possibly in future work) compare the results of the present formulation with that of a full finite volume approach, that should not be difficult to implement in their framework.
b) Nothing is said on the well balancing properties of the scheme and on the spurious velocities that may arise in an atmosphere at rest with a large mountain at the bottom; an explicit discussion of this point and a short description of the outcome of one such test should be included. Also related to this, the method described in the paper does not appear to require the use of a reference profile: for clarity, the authors might state explicitly if this is not the case.
c) From the sentence 'For each cell two cell-centered values of each of the three components of the cartesian velocity vector are computed and transported with the above advection scheme for a cell-centered scalar value.' (page 4470) it would appear that the proposed approach requires twice the computational effort than an approach based directly on staggered control volumes. The authors should clarify this point and, should this be really the case, justify this rather expensive choice with respect to more straight- forward approaches based on the use of a staggered control volume. Furthermore, it is important to understand if and to which extent the flux limiters in the momentum equations are acting just to suppress some inaccuracy related to the proposed shaved cell approach; the typical values of the flux limiter around the orography should be re- ported, to understand whether the method is mostly reverting to first order upwind or not.

b) **Our answer:**
Details on numerics a) To approximate the pressure gradient at the interface of two grid cells with only the pressure values of the two grid cells there is some freedom in choosing the grid size. Whereas in Adcroft et al. the grid size is chosen to preserve energy in their model. We follow Ng et al. (2009): Y.-T. Ng, H. Chen, C. Min and F. Gibou (2009): Guidelines for Poisson Solvers on Irregular Domains with Dirichlet

Boundary Conditions Using the Ghost Fluid Method. Journal of Scientific Computing, 41, 300-320.
We have implemented both versions in our code and found that the second one is more suitable to simulate flows in hydrostatic balance.
b) A reference profile was used in earlier versions and is now discarded. See the discussion above.
c) Yes, for advection the amount of computation is doubled for the three velocity com- ponents but this is negligible compared to the number of transported scalars in sophis- ticated microphysical schemes. This approach avoids separate advection routines for the momentum components. We have also implemented a version with only on cell centered velocity components for advection and back interpolation, which seems to be more diffusive.

c) **Changes in the manuscript**
We added some paragraphs in 2.2.3 to address these points.

4)

a) **Referee comment:**
As far as I know, one of the best known attempts to apply finite volume concepts to describe orography/topography in environmental models is
A. Adcroft, C. Hill, and J. Marshall, Representation of topography by shaved cells in a height coordinate ocean model, Monthly Weather Rev. 125, 2293 (1997).
The authors ignore this seminal paper in their reference list. Furthermore, in the paper by Adcroft et al a number of idealized cases for advection over orography/bathymetry are proposed, that would allow to assess the accuracy of the proposed method close to the bottom. The authors should consider performing one of such tests, in particular for the purpose of assessing which kind of accuracy is to be expected in the shaved cells, assuming that no theoretical argument to estimate the convergence order at the bottom is available. A method that reverts to first order in the lowermost cells could introduce excessive numerical diffusion in the lowest layers, thus making the proposed approach not extremely useful in practice. The authors should discuss this issue and present new results that clarify the properties of their method in this respect.

b) **Our answer:**
Reference paper to finite volume concepts
We are aware of this paper and will cite it.

c) **Changes in the manuscript**
We cited this and another important paper in the last paragraph of 2.2.3.
We have added other test examples for accuracy tests in Sec. 4.

5)

a) **Referee comment:**
Since the main novelty of the proposed approach is the finite volume treatment of the orography, this technique should be tested in a much more systematic and quantitative way on idealized benchmarks, where its accuracy can be assessed much more clearly in comparison with similar or alternative approaches. In particular,

besides trying an advection test as suggested in point 4, the following remarks should be addressed:
a) Concerning the cold bubble Straka test, the statement that 'These values and the contour field agree well with the results from the literature' is debatable at best. A number of different methods in recently published papers have been used to simulate this benchmark, I quote more or less at random
Klemp, Joseph B., William C. Skamarock, and Jimy Dudhia. "Conservative split-explicit time integration methods for the compressible nonhydrostatic equations." Monthly Weather Review 135.8 (2007): 2897-2913.
Giraldo, Francis X., Marco Restelli, and Matthias Läuter. "Semi-implicit formulations of the Navier-Stokes equations: application to nonhydrostatic atmospheric modeling." SIAM Journal on Scientific Computing 32.6 (2010): 3394-3425.
Norman, Matthew R., Ramachandran D. Nair, and Fredrick HM Semazzi. "A low communication and large time step explicit finite-volume solver for non-hydrostatic atmospheric dynamics." Journal of Computational Physics 230.4 (2011): 1567-1584.
In all these papers, different methods at different resolutions, either lower or higher than the one used by the authors, consistently give a front position at t=900 s that is on the left of the 15 km mark, while the solution in the present paper is well beyond that. Furthermore, a much larger spacing is employed between subsequent contour levels (2K rather than 1 K or even 0.25 K in the referenced papers). The authors should address this discrepancy and try to explain it, as well as replacing the plot with one using a contour spacing comparable to that used in the literature.
a) The only idealized test with orography concerns an orographic obstacle that does not go beyond the first model layer. This is hardly a tough test for a shaved cell method, in the sense that even rather inaccurate approaches may pass such a test. At least one lee wave test should be run in which a mountain profile is used that intersects several grid layers (several such tests are presented in the literature). Details of the flow around the obstacle should be analyzed and a quantitative comparison in terms of analytically predictable quantities (vertical momentum flux) should be presented.
b) If moist idealizd tests are to be included, at least one of them should concern lee waves with moisture, in order to compare with results like those presented e.g. in Durran, Dale R., and Joseph B. Klemp. "A compressible model for the simulation of moist mountain waves." Monthly Weather Review 111.12 (1983): 2341-2361.
Miglietta, M. M., and A. Buzzi. "A numerical study of moist stratified flow regimes over isolated topography." Quarterly Journal of the Royal Meteorological Society 130.600 (2004): 1749-1770.
Miglietta, M. M., and R. Rotunno. "Simulations of moist nearly neutral flow over a ridge." Journal of the atmospheric sciences 62.5 (2005): 1410-1427.
Also in this case, a mountain profile intersecting more than one grid layer should be used.
c) Concerning the more realistic test, I can hardly assess its meaningfulness until the previous issues concerning the numerical methods are cleared. However, should the authors want to include a more realistic test, I strongly recommend that they chose one on which other shaved cell approaches have already been applied, so as to allow for a comparison with alternative techniques.

b) **Our answer:**
Idealized benchmarks
In the papers by Giraldo et al. and Norman et al. the initial bubble is different from the one de- scribed by Straka et al. Compare with the test suite of Skamarock: http://www2.mmm.ucar.edu/projects/srnwp_tests/density/density.html. We have also tried this initial perturbation and get similar results and get a front that is on the left side of the 15 km mark (see attached figure). The actual contour spacing in Fig. 5 is1K.The2Kinthecaptionisatypoandwillbecorrected. Itmightalsobeabitof misleading to show the 300 K contour line. In most figures in the literature they start 299.5 K and 1 K steps.
a) + b) We will add a moist test case with steeper orography and compare our results with the ones from the following work: Kunz, M., Wassermann, S. (2011): Sensitivity of flow dynamics and orographic precipitation to changing ambient conditions in idealized model simulations. Meteorologische Zeitschrift, 20(2), 199–215.

c) **Changes in the manuscript**
The test case section has been reworked completely.

6)

a) **Referee comment:**
*Two different semi-implicit solvers are described in section 2.3. Firstly, it is not entirely clear whether the application of these methods to the Euler equations with gravity is a novel development of this paper or was proposed already in Jebens et al 2011. The authors should clarify this point. They should also make clear which of the approaches is used in the numerical tests and whether any significant difference in accuracy or performance is noticed between the two. The description of the two approaches is so intermingled that it is difficult for the reader to sort out what is actually done in each case. The authors should try to streamline the description of each variant. From the point of view of the linear solvers employed, it is unclear what the 'conjugate gradient (CG)-like methods' referred to on page 4473 exactly are (Bi-CGSTAB? GMR?) and what is an estimate of the resulting computational cost. The authors should provide e.g. some information on the average number of iterations as a function of the typical Courant numbers |c|*dt/dx and |u|*dt/dx, where c is the speed of sound and u the flow velocity. Finally, it would be interesting if the authors could comment on the possibility*
of recovering within their framework the discretization of the pseudo-incompressible approximation of the Euler equations, as done e.g. in
T.Benacchio, W.P. O'Neill, R. Klein A blended soundproof-to-compressible numerical model for small to mesoscale atmospheric dynamics,Monthly Weather Review 2014 doi: http://dx.doi.org/10.1175/MWR-D-13-00384.1

b) **Our answer:**
Semi-implicit solvers Rosenbrock-W-methods are a special class of linearly implicit solvers. In these methods the compressible system is handled as a whole. Their application in numerical weather prediction is already described in an Oberwolfach Report in Knoth (2006). Since the approximated Jacobian can be "arbitrarily" chosen, different types of explicitnesses can be reached. Especially two types of "pressure" solvers result from this approach where for most applications the simpler approach is

sufficient. Both iterative linear solvers, BiCGStab and GMRES, are standard iterative methods and work well with suitable preconditioners. The number of iterations for the two iterative methods are problem dependent. They increase with increasing time step and are usually in the range of 2 to 5 iterations. Unfortunately the iterative solver for the sound part (pressure solver) do not scale well in case of a parallel use of the model. The parallelization is not described in this paper and will postponed to further special topics of the model. There is no connection to the work by Klein et al.

c) **Changes in the manuscript**
We have added some further comments regarding the description of the methods and re-formulated some sentences in Sec. 2.3 for better clarification. Also, notation issues were corrected. We now mention the used iterative solvers and give a range of the number of iterations.

Anonymous Referee #2

7)

a) **Referee comment:**
The introduction focusses on the cut-cell formulation of the model and the advantages of this for atmospheric research, however the review of existing cut cell models here is a bit patch and missed a number of relevant works including Adcroft et al (1997), Steppler et al (2006), Yamazaki and Satomura (2008, 2010, 2012) and Good et al (2014). No reference is included for the numerical schemes mentioned (Rosenbrock time integration).

b) **Our answer:**
We are aware of these articles and will cite them. A Rosenbrock citation with application to the incompressible Navier-Stokes equation is included. We mention an old own paper with respect to this subject.

c) **Changes in the manuscript**
All relevant articles are now cited in the introduction.
Citations for Rosenbrock methods are now added in Sec. 2.3 and the introduction.

8)

a) **Referee comment:**
The model presented is not a new model, but is a development of an existing model (used for CFD of flow round buildings) for atmospheric models. Given this, it is rather odd that there are no references for previous versions of the model or a clear statement of what is new here. There is just one mention of recent use of the model for urban environments.

b) **Our answer:**
This is the first description of the model ASAM in the literature. Part of the numerics and some model application are already described in the literature.

c) **Changes in the manuscript**
We added all works where ASAM was involved in the introduction and also give some

more information about recent applications of the model.

9)

a) **Referee comment:**
You present the Euler equations in (1)-(3), but later on you include a LES sub-grid model so there should be some form of source term / Reynolds-stress term in (2). You also mention a "constant physical viscosity" in the cold bubble test? Where does this fit into the Euler equations?

b) **Our answer:**
We will add a term representing some type of Reynolds stress, depending on the application.

c) **Changes in the manuscript**
We added subgrid scale terms in the Euler equations (1)-(3) for momentum and scalars.

10)

a) **Referee comment:**
I realise notation can be tricky in describing numerical schemes, but there is a problem throughout of using lots of notation, not all of which is clear and not all of which is defined. As an example on page 4468 you define F UL as the area of the U face, but this is confusing with F × U . Perhaps notation like FU R would be clearer? Lower down the page you also talk about UFL but don't say what this is, nor do you explicitly define φL and VC. I infer that the subscript means that variable on the L / R face or the cell centres to the L / R? There also appears to be dupliation of notation (e.g. φ is a scalar variable in the preceding sections then suddenly in equation (9) φ is the limiter. This whole section needs checking. Please make sure all variables are clearly defined throughout. Other examples include kj , βij and γij in equation (16)

b) **Our answer:**
We did some notation changes and tried to improve the representation of the main cut cell.

c) **Changes in the manuscript**
Notation should now be consistent.

11)

a) **Referee comment:**
How do your scheme for interpolating values onto the faces work near the boundaries where some adjacent cell values are not defined? How do you cope with faces which are only partial faces? Is interpolation from the cell values the most appropriate way of dealing with this? I found that your explanation didn't seem to really address how you handle the cut cells, which seems to be the critical bit of the whole model description.

b) **Our answer:**
Most of the formulas describing the spatial discretization involve the cut cell information, like free face area and free volume. No other tricks are used in the code.

c) **Changes in the manuscript**
We reworked Sec. 2.2 for clarification.

12)

a) **Referee comment:**
I was confused by equation (13). It is discontinuous at FUL = FUR. I assume there is a mistake?

b) **Our answer:**
There was a typo and we corrected it.

c) **Changes in the manuscript**
See b)

13)

a) **Referee comment:**
Given the focus on the cut cell capabilities of the model, it is rather important to see how the physics parameterisations such as the sub-grid scale model and the surface fluxes deal with this. Despite the detailed descriptions, there is relatively little detail or testing of this point. In particular, how does the interpolation in the cut cells affect the accuracy and conservative properties of the model? Some tests to prove this would be useful.

b) **Our answer:**
Total mass and the mass of water substances are conserved by construction. Regarding other properties like total energy we do not expect any exact conservation. We will add a bubble experiment where the bubble interacts with the topography to demonstrate "good" energy conservation. We have added a short note how surface fluxes are distributed in the vicinity of a cut cells.

c) **Changes in the manuscript**
Tests regarding energy conservation are done in Sec. 4. A flux distribution technique has been added (Sec. 2.2.4 + Fig. 5).

14)

a) **Referee comment:**
Overall I found the description of the model microphysics rather detailed. It ap- pears to me that much of this is not particularly novel. I would suggest instead focussing the paper on properly testing the dry-dynamics cut cell aspects of the model. Some aspects (e.g. the surface fluxes and soil model) are not even used in the test cases presented here.

b) **Our answer:**
We will add the valley example from the inter-comparison paper by Schmidli et al.

(2011), which includes part of the soil-vegetation model to compute surface fluxes and turbulence parameterization amongst others.

c) **Changes in the manuscript**
Due to the brevity of time, the valley test case could not be added to our rework of the test cases section. See also 1c).

15)

a) **Referee comment:**
The test cases are useful, but I would question whether these are the most ap- propriate test cases for a cut-cell model. The cold bubble is a common test case, but does not of itself test the cut-cells since the surface is flat. (Of course, this depends exactly where the surface is with respect to the grid, but there are no details given of this.) I found the comparison with the previous work rather super- ficial. There are no figures given from the original Straka et al paper, for direct comparison. Looking at this paper there seems to be some differences, with ad- dition contours, despite the fact that the contour interval is 2K here (compared to 1K in the original paper). I would also like to see some values for maximum / minimum theta perturbations to compare with the original paper. This is also a useful test of the monotonicity of the scheme. The description of the setup mentions a fixed physical viscosity, however this is the first mention of this - it does not appear in the equation set given above. Where does the value come from? The original Straka paper used a fixed K, but here it appears that you have a turbulence model instead? Or is the turbulence model turned off in this case? If so why?

b) **Our answer:**
We have changed the number and field of values for the contour plot. The viscosity is switched on and is taken as a part of the viscous tensor with the prescribed fixed diffusion coefficient. See also our new energy test.

c) **Changes in the manuscript**
We overall improved the quality of the plots. With the test cases section rework, we addressed all issues you mentioned.

16)

a) **Referee comment:**
Moist bubble. Is equation (80) the perturbation in ϑ or ϑe - it's not clear. I assume this is only for $L \leq 1$? Again this problem does not test the cut cells at all. You might consider trying the moist bubble over a hill as done in Good et al (2014). With a cut cell model there should be negligible difference between bubble ascent with and without a hill. This is a useful sanity check, although still not a tough test of the cut cells. There is an additional test case with a uniform speed of $U = 20ms{-}1$. How does this square with the (presumably) no-slip boundary conditions? I could find no proper discussion of the lower boundary conditions on velocity in the model. This is another important and tricky aspect to get right in a cut cell model so needs discussion.

b) **Our answer:**
We follow the description of the original paper of Bryan and Fritsch (2002) in full

detail. In your recommended moist test case with cut cells the main flow evolves in the undisturbed part of the domain. Hence we see no benefit from this exercise. To "shorten" the paper we can also remove this test case.

c) **Changes in the manuscript**
We are aware of the Good et al. (2014) paper and cited it previously. However, we think that there should be some interaction with the orography. For this reason, we presented the 'Zeppelin' test case with the moist bubble of Bryan and Fritsch (2002) in Sec. 4.2.

17)

a) **Referee comment:**
The mountain wave case is a more useful standard test case for the cut cell model, however there was only a single paragraph presenting and discussion of this case with no real quantitative comparison with other studies. I would certainly expand on this. Why not compare directly with the analytical solution in Schar et al?

b) **Our answer:**
Note that we have added a second mountain test case, now with moisture and a steeper hill.

c) **Changes in the manuscript**
With the 3D mountain overflow case in a moist atmosphere presented in Sec. 4.4, we removed the Schaer test case.

18)

a) **Referee comment:**
The final test case over Barbados is a test in the sense that it will check the model runs with real terrain and does something sensible looking, but there are no observations, analytical solutions or equivalent simulations with other models to compare against. I would suggest leaving this test out of a preliminary paper, and making a more thorough microphysics test, comparing with observations or other models, the subject of an second paper. I have a couple of other questions pertinant to this test which need addressing too. The description says a stretched grid in the vertical is used. Does this mean the vertical resolution at the surface is less at altitude over the island? This is a problem for cut cell models and needs discussion. A test to look at the effect of this would be good (perhaps c.f. a terrain following model?) The initial profile used with constant N, a log wind up to 300m, with constant wind above, and a humidity inversion. This doesn't seem dynamically consistent. A plot of the profiles (particularly of humidity) would be useful. How long does the model take to reach a balanced state? Are the results you show in this state? Where do the specified values of z0 come from? They seem very small over the ocean and quite large over land. Incidentally, this is the first mention of z0 as far as I can see. How is it used in the model (see previous point about lower boundary conditions)?

b) **Our answer:**
We would like to keep this "real" test case in the paper, but in a re-worked and condensed version. It will be a simple sensitivity study with 1) a flat island surface and

2) the real island orography to demonstrate terrain effects and that the inclusion of real topographical structures works well. Extensive analyses regarding this topic will be postponed to future papers. The authors think that this is still a good and meaningful test case since it shows how the numerics, the discretization and the model physics interact (even if there is no possibility to compare with other studies [yet]). For this test case, we will now use the BOMEX (Barbados Oceanographic and Meteorological Experiment) profile, which is extensively described in the literature, cf. Siebesma et al. (2003). It also includes a realistic wind profile. Surface drag will be directly modeled by the momemtum flux parameterization. Issues with variable vertical grid spacing will be addressed in the valley example (cf. point 8).

c) **Changes in the manuscript**
We agree that it is not consistent in the framework of our paper design to present this application. Thus, we removed the “Barbados” section in order to focus on idealized test cases that are reproducible and test conservation properties and model accuracy.

H. Weller (topical editor)

19)

a) **Editor comment:**
You call this an "All Scale Atmospheric Model". The model uses a logically rectangular grid, so how are you planning to cope with the pole problem? Lat-lon grid? In order to call the model all scale, you should address this point.

b) **Our answer:**
The model can also be used with spherical or cylindrical grids. The stability problems with the grid convergence in special points (the pole problem) in both grids are handled through the implicit time integration both for advection an the yet faster gravity and acoustic waves.

c) **Changes in the manuscript**
We mention it in our new introduction and added Section 2.4, where further grids are described.

20)

a) **Editor comment:**
I wouldn’t call the metric terms associated with terrain following coordinates "artificial forces".

b) **Our answer:**
We will change this imprecise formulation.

c) **Changes in the manuscript**
“… and the numerical pressure derivative in the vicinity a structure is zero, which is not the case in terrain-following coordinate systems due to the slope of the lowest cells …”

21)

a) **Editor comment:**
What is a "logically orthogonal rectangular grid"?

b) **Our answer:**
A logically rectangular grid has the same logical structure as a regular Cartesian grid. Especially it has the same number of nodes, faces etc. and the same neighbor relations.

c) Changes in the manuscript
We added a note in brackets.

22)

a) **Editor comment:**
The treatment of the non-linear advection term is not clear (page 4470).

b) **Our answer:**
This issue was already mentioned by the first referee. We changed this in our manuscript for more clarification.

c) **Changes in the manuscript**
See 2) and 3).