

Geosci. Model Dev. Discuss., author comment AC4
<https://doi.org/10.5194/gmd-2021-255-AC4>, 2022
© Author(s) 2022. This work is distributed under
the Creative Commons Attribution 4.0 License.

Reply on RC2

Ivo Suter et al.

Author comment on "uDALES 1.0: a large-eddy simulation model for urban environments"
by Ivo Suter et al., Geosci. Model Dev. Discuss.,
<https://doi.org/10.5194/gmd-2021-255-AC4>, 2022

We thank reviewer #2 for their constructive comments which have made the paper more accessible and rigorous.

The main changes made in this revision are:

- Further explanation on the technical elements of the paper;
- Detailed validation metrics for the DAPPLE case;
- Substantial improvement of the East-side demo:
 - Reran the experiment with the use of a driver simulation rather than a periodic simulation in order to prevent heating up of the air;
 - This allows the simulation time to be 8 hours rather than one hour.
- Fluxes are now presented conforming to atmospheric convention.
- A thorough check of all simulation cases in the repository, with updated input and run-options

We respond to the specific points made by the reviewer below.

Comment 1: The novelty of this open-source software should be further strengthened. Specifically, compared to OpenFoam and PALM, what is the advantage of this software? Does it also allow modules to be integrated by other users?

Authors reply: There are many LES solvers capable of solving flow around obstacles/buildings, such as the open-source code OpenFoam. However, they generally do not include many critical components to determine the urban climate, e.g. humidity or a surface energy balance.

The goals of uDALES and PALM-4U overlap to a large extent. Both models aim to incorporate a large number of processes that determine the urban climate. As such there is no direct advantage over PALM, rather uDALES is an alternative. Compared to global or regional models, there still seems to be a lack of models on an urban scale and we believe having multiple open-source options will benefit everybody. We encourage contributions by other users, and uDALES is set up in a modular fashion that makes it relatively straightforward to add custom modules. We have extended the Introduction to emphasise these points.

Comment 2: What extension have you added based on DALES? This should also be

clearly stated in this paper.

Authors reply: These mainly pertain to the urban surface. We have added a list of differences with DALES at the beginning of the Model description section.

Comment 3: The computational time and computational RAM or CPU requirement should also introduce in this paper, as well as the data communication process.

Authors reply: A typical use case of 384^3 points at 2 m resolution using a fixed timestep of 1s took about half an hour to simulate 10 minutes using 196 cores. The parallelisation is based on DALES, and we refer to that paper for a more detailed description of the parallelisation. We mention the use of MPI in section '2.2 Method of Solution' and the computational requirements in sections '4.1.1 Simulation set-up' and '4.3 Eastside demo'.

Comment 4: In equation (2), there is a notation θ_u ; and in equation (3), there is another notation θ_v . Are they the same notation? If so, please make it consistent, otherwise, please use different form to avoid misunderstanding.

Authors reply: Equation 2 and equation 4 both use the same notation θ_v (θ_v). Equation 3 does not include a θ . In the text at 191 however we mistakenly used a cursive 'v' (θ_v).

Comment 5: For the schematic diagram in Figure 1, I suggest the authors could add some symbolic simulation obstacles in simulation section. I guess it could be the latter part in Figure 1b?

Authors reply: We added some symbolic obstacles to the figure.

Comment 6: For section 2.3.2, could you please explain in more detail about the numerical settings for lateral boundary condition, especially the inflow-outflow condition? Does it mean that the upper part of the simulation domain is used to generate flow turbulence and set as periodic, then feed into the latter part? Also, what is a "run-up" region in figure 1c?

Authors reply: When using inflow/outflow boundary conditions, the y direction is periodic so is still solved using FFT, but x and z are solved using cyclic reduction. The inflow boundary condition for pressure is Neumann and the outflow is Dirichlet. The bottom and top are both Neumann boundary conditions. We have clarified this in section '2.2 Method of Solution'.

The driver simulation typically uses periodic boundary conditions and is run beforehand. The velocity and scalar fields on an outflow plane are saved. These are then loaded into the target simulation with inflow-outflow boundary conditions. The outlet boundary condition is convective, using the vertically-averaged velocity profile as the outflow velocity. The run-up region is an optional region in which the flow can develop. We have added additional information in section '2.4.2 Lateral boundaries'.

Comment 7: Page 21, What do you mean by stating the following sentence "All processors know about the facet properties and calculate the local wall fluxes according to the state of the local fluid cells. For the facet energy balance the processor average has to be determined."? Could you please give more details for clearer understanding? Does it mean the data will be summarized in one processor via MPI and then averaged or other operation and send to other processors?

Authors reply: We agree that this was a confusing passage. The surface energy balance calculations are performed on a single process, and the heat flux data from the facets

needs to be gathered from all the other processes. Once the surface energy balance has been calculated, the new facet temperatures are distributed to all other processes so the wall-functions can be updated. We have clarified this in section '3.7 Integration into LES'.

Comment 8: Refer to the previous question, does this data communication need to be done every time iteration? If so, how much time does it cost, maybe a rough comparison between data communication and computation in each time step?

Authors reply: Yes, the summing of the heat fluxes over the facets and their time integration is currently done at every LES time-step. These two steps could potentially be separated where the time integration is done locally and the summing is only done every energy balance time-step. However, the impact on performance will be minimal, since it is two calls to `MPI_GATHER(..,MPI_SUM,..)` of maximum a few MB of data, followed by two broadcasts. We added a statement that the computations related to the facets are orders of magnitudes lower than those related to the fluid cells in section '3.7 Integration into LES'.

Comment 9: In table 1, the grid size for main simulation is 450X400X100, and the domain size is 900X800X200. I do not think that there are only 2 grid points in x, y and z direction. So, I think the grid size should be grid number or grid points. Please revise accordingly.

Authors reply: Added a row 'cell size [m]: 2x2x2' so it is clearer that 450x400x100 is the dimensions in number of grid points and 900x800x200 is the dimensions in meters.

Comment 10: In figure 7, could you please indicate clearly about the position R?

Authors reply: We added a detailed view of the measurement points in the intersection to Figure 7. The measurement point R was further renamed to P1.

Comment 11: The test case of wind tunnel experiment results of the DAPPLE project should be briefly introduced in Section 4.2. Or relevant materials should be provided in supplements.

Authors reply: The sections '4.1 Validation' and '4.1.1. Simulation set-up' were revised and expanded to now include a more detailed description of the study case set-up (urban geometry etc.) and wind-tunnel experiment, including their methods of data acquisition (Carpentieri et al., 2009, 2010, 2012). Section '4.1.2 Results' now also includes quantitative comparison between simulations and measurements using several validation metrics, with the relevant wind-tunnel data available in the supplement.

Comment 12: What is the test case in Figure 11 (surface energy balance test case)? This should be carefully described in the paper.

Authors reply: We describe now that we use a case where we try to maintain parameters describing the building morphology between the two-dimensional MTEB model and the three-dimensional uDALES. MTEB uses a street canyon representation for the built environment and e.g. the building width in the MTEB should correspond to the building area in uDALES. Added to '4.2 Verification of surface energy balance'.

Comment 13: In section 4.3, have you validated this case with on-site measurements? I think it will be more convincing if the authors can validate their codes with real time measurement.

Authors reply: Unfortunately, there are no measurements available. We have to leave the comparison with an instrumented green roof to a future study.

Comment 14: In section 4, have you done grid sensitivity test on these case studies? As I understand, the filter in LES simulation in this paper is implicit and based on grid size. So, the grid size should be also important for the simulation quality. Perhaps, the authors can also add turbulent kinetic energy analysis in these test cases analysis.

Authors reply: For the DAPPLE validation simulation, we analysed the subgrid fluxes and they do not exceed 4.5% of the total flux anywhere except for in the lowermost cells, indicating that the resolution is sufficient to resolve the majority of the energetic turbulent scales of the urban flow field. As always there is a computational cost associated with finer resolutions and we found 2m to be adequate for the problem. We now mention that the subgrid-scale fluxes are small in '4.1.1 Simulation set-up'

Comment 15: How is the grid arrangement of this software? Does it only allow hexagon grid? Does the grid need to be uniform in x, y and z direction?

Authors reply: uDALES is based on a Cartesian Arakawa C-grid (Arakawa and Lamb, 1977). The grid can be stretched in the z-direction when using periodic boundary conditions, and in the x- and z-directions when using inflow-outflow boundary conditions. We now mention this more clearly in section '2.2 Method of Solution' and section '2.3 Immersed boundary method'

Comment 16: Page 2, In the following sentence, I think "transition" should be "transit". "The increased likelihood of extreme weather events due to climate change (IPCC, 2014) and the need to transition to a less".

Authors reply: After consulting the Oxford Dictionary we believe 'transition' is correct: 'transition, verb: to change or to make something change from one state or condition to another' whereas: 'transit, verb: (something) to pass across or through an area'

Comment 17: Page 3, please check the citation format in the text "require turbulence parameterisations for the full range of active scales in the flow field Blocken (2015)." Please also do this for the whole paper and revise accordingly.

Authors reply: We checked the entire paper for citations which were not enclosed in parentheses, but should be; and vice versa.

Comment 18: Page 3, please check the following sentence and revise "Popular codes for RANS are any of the commercial CFD packages (Fluent, ANSYS CFX, COMSOL etc), although there are also open source alternatives (e.g. OpenFoam)."

Authors reply: We rephrased the sentence, it now reads: 'There are many RANS codes available, including commercial CFD packages (Fluent, ANSYS CFX, COMSOL etc) and open source alternatives such as e.g. OpenFoam.'

Comment 19: Page 3, I think "large turbulent scales" rather than "bulk of the turbulent scales" in the following sentence "Large-Eddy simulation (LES) tools explicitly resolve the bulk of the turbulent scales in the flow,"

Authors reply: We rephrased the sentence, it now reads: 'Large-Eddy simulation (LES) tools explicitly resolves the large turbulent scales of the flow that contain the majority of the turbulent energy, and are therefore less reliant on turbulence models than RANS'