

***Interactive comment on “A comparison of a two-dimensional depth averaged flow model and a three-dimensional RANS model for predicting tsunami inundation and fluid forces” by Xinsheng Qin et al.***

**Xinsheng Qin et al.**

[xsqin@uw.edu](mailto:xsqin@uw.edu)

Received and published: 19 July 2018

The authors would like to thank the reviewer for time and the insightful comments. We have incorporated these comments into the revised manuscript and hope that we have addressed any concerns. Specific responses to review comments are shown below.

1. As mentioned above, CFD of tsunamis are relatively rare. As this is much of the novelty of the present work, a more thorough literature review on this general topic would seem warranted, as several seemingly relevant papers are not cited. Such works

C1

seemingly include: Biscarini (2010), Montagna et al. (2011), Larsen et al. (2017), as well as Aniel-Quiroga et al. (2018).

We have added those to the introduction section. Aniel-Quiroga et al. (2018) (“Stability of rubble-mound breakwaters under tsunami first impact and overflow based on laboratory experiments”) is an experiment study that does not include any numerical modeling of tsunamis so we opted not to include that.

2. p. 5:  $B(x,y)$  is ambiguously defined as the topography. Does this mean the bed elevation? Please clarify.

This has been clarified in the manuscript.

3. p. 9: It is stated that  $z$  is perpendicular to the flume bottom. Would it not be simpler to state that  $z$  is vertical?

This has been updated in the manuscript.

4. p. 9: Discussing mesh resolution strictly in dimensional terms gives little physical meaning. Please also add discussion in terms of wall units,  $z \rightarrow z = z^* U_f / \nu$ , where  $U_f$  is the friction velocity and  $\nu$  the kinematic fluid viscosity.

The mesh sizes in the unit of wall units have been added for both models in the manuscript in page 9.

5. p. 9: I am not convinced that simply making  $B(x,y)$  very large properly simulates a column. How exactly has this been tested? Why should the vertical column wall be modelled differently than other vertical walls?

We chose to model a column this way since we would like to make use of GeoClaw’s ability of handling dry and wet cells in tsunami inundation. GeoClaw has been shown to be able to model the movement of shorelines well [1-3], during which computational cells can switch between dry and wet states. Any edge between a cell that is wet and a cell that stays dry in a time step is handled with reflecting boundary conditions and

C2

acts as a wall. So setting the topography in the column high enough that it never gets wet is an easy way to impose wall boundary conditions around it.

The vertical walls of the flume are boundaries of the computational domain, which requires boundary conditions (in this case, they all have reflecting wall boundary conditions). In theory, the vertical column can be modeled in the same way with a modification of the code to introduce interior boundaries, but this is not necessary since the same conditions are already imposed at the interface between wet and dry cells

[1] George, D., & LeVeque, R. J. (2006). Finite Volume Methods and Adaptive Refinement for Tsunami Propagation and Inundation. PhD Thesis, 24(5), 319–328. [2] Berger, M. J., George, D. L., LeVeque, R. J., & Mandli, K. T. (2011). The GeoClaw software for depth-averaged flows with adaptive refinement. Advances in Water Resources, 34(9), 1195–1206. [3] LeVeque, R. J., George, D. L., & Berger, M. J. (2011). Tsunami modelling with adaptively refined finite volume methods. Acta Numerica, 20, 211–289.

6. Please add axes with labels to Figure 1, this will greatly improve clarity.

The x and y axes have been added.

7. Forces are estimated using a drag coefficient in Eq. 20. Why is the more general Morrison equation not used?

The Morison equation is a semi-empirical equation for the inline force on a body in oscillatory flow and is widely used in computing wave loads on offshore structures like cylindrical legs of an ocean platform. It is written as  $F_{\text{morison}} = F_{\text{inertial}} + F_{\text{drag}} = \rho * C_m * V * du/dt + 0.5 * \rho * C_d * A * u * |u|$ , where  $u = u(t)$  is free stream velocity and usually known from wave theory, depending on which type of wave is assumed (Linear, Stokes 3rd order, or Stokes 5th order etc.). The equation also assumes that the flow acceleration is more or less uniform at the location of the body. For instance, for a vertical cylinder in surface gravity waves this requires that the diameter of the cylinder is much smaller than the wavelength. We believe these typical

### C3

application scenarios for the Morison equation are not applicable to our problem. For example, assuming we have a building that's 1m x 1m x 1m, if we sample velocity of the fluids at a point that's 0.5 m in front of it and 0.5 m above the ground, versus at a point that's 1 m in front of the building and 0.8 m above the ground, and use time histories of these two velocities to compute the inertial force and drag force in the equation, the results can be very different for the cases we present.

Note that in the manuscript, we don't have this problem when drag force is computed, since we used a case where the building was removed and the velocity was sampled at the center of the building. The history of this velocity cannot be used to compute the inertial force in Morison equation since there is no building resisting the flow thus  $du/dt > 0$  during most of the time (until the peak has passed the location of interest). Again, we believe although this semi-empirical equation is used conventionally in ocean and petroleum engineering, it is not suitable in current scenario.

In addition to the discussion above, using drag coefficient to estimate the hydrodynamic forces that contributes to tsunami loads on structures is what's recommended by FEMA P-646 [1] as well as by the chapter for tsunami loads and effect for coastal structures in the latest ASCE 7-16 [2]. These practices have been used for the tsunami hazard study community and recently, applied to the design of the first tsunami vertical evacuation structure in the United States [3].

[1] Applied Technology Council: Guidelines for Design of Structures for Vertical Evacuation from Tsunamis. Second Edition (FEMA P-646), FEMA P-646 Publication, [https://doi.org/10.1061/40978\(313\)7](https://doi.org/10.1061/40978(313)7), 2012 [2] American Society of Civil Engineers (ASCE): Minimum Design Loads for Buildings and Other Structures, Standard ASCE/SEI 7-16, 2016 [3] Ash, C.: Design of a Tsunami Vertical Evacuation Refuge Structure in Westport, Washington, in: Structures Congress 2015, pp. 1530–1537, 2015

8. p. 15: The function  $s(t)$  is given, but without specifying parameters  $A$  and  $\beta$ ,

### C4

hence the reader is given no information regarding the duration. Please clearly define these parameters. Sufficient information must always be given such that scientific work is repeatable. Also, this equation is repeated as Eq. 24. To improve efficiency, please give this an equation number on first use, and avoid repetition of equations.

This has been addressed accordingly in the manuscript.

9. p. 18: It is stated that zero fluctuations in the along shore directions are assumed. This makes no sense - turbulence is always three dimensional, and there is no physical situation where such an assumption is justified, and Eq. 23 is not a proper estimation of  $k$ . The turbulent kinetic energy  $k$  can be approximated by one component, but this should involve a factor 1.25 (see e.g. Scott et al. 2005) rather than 0.5 in Eq. 23. Please correct this and revise accordingly.

We have corrected this in the manuscript. We did not apply the factor 1.25 before since the same approach is used in [1].

[1] Lin, P. and Liu, P. L. F.: A numerical study of breaking waves in the surf zone, *Journal of Fluid Mechanics*, 359, 239–264, <https://doi.org/10.1017/S002211209700846X>, 1998.

10. Table 1: `fixedValue` is indicated for the velocities - which value? (Presumably this is zero, but this certainly needs to be clarified).

Yes, for the `fixedValue` boundary condition, a constant value of 0 is used for the model in this study. This has been added to the manuscript.

Please also note the supplement to this comment:

<https://www.nat-hazards-earth-syst-sci-discuss.net/nhess-2018-150/nhess-2018-150-AC2-supplement.pdf>

---

Interactive comment on *Nat. Hazards Earth Syst. Sci. Discuss.*, <https://doi.org/10.5194/nhess-2018-150>, 2018.