Response to Reviewer #1 for Manuscript "High-Fidelity Modeling of Turbulent Mixing and Basal Melting in Seawater Intrusion Under Grounded Ice" by Mamer, Robel, Lai, Wilson, and Washam

General comments:

The authors aim to tackle a fascinating problem of practical importance. Melting near grounding lines is thought to have much more impact on glacier dynamics than melting anywhere else, yet observations are limited, and high-fidelity simulations are lacking. Thus, the work is highly novel and has significant potential for improving our understanding of ice-ocean interactions that have a high impact on climate dynamics. However, I am not sure that the ANSYS Fluent RANS solver used is appropriate for this problem. In fact, I have never seen the ANSYS Fluent RANS solver applied to environmental flows as complex as in this work. This does not mean that it cannot work, but that significant effort should be devoted to validating the code. As the governing equations solved by the codes are not clearly presented or discussed (in particular, the boundary conditions, which are yet key to evaluating the melting dynamics), I have found it difficult to assess the appropriateness of the mathematical/numerical model. I am particularly concerned with the modelling of the ice-ocean boundary: there are no salt constraints and melt is said to be activated in a way that I did not find correctly physically motivated. In the unfortunate case that the code cannot in fact solve the exact problem at hand (with temperature and salt stratification coupled through a phase change boundary) I would suggest that the authors reformulate the problem as a list of hypotheses--inspired from the full problem--and testable with simulations of a modified/simpler model (but mathematically transparant), which could be the reduced thermal driving model discussed below (point 2).

We thank the reviewer for their thorough comments on this manuscript. We have identified this reviewer's primary concerns to be:

1. The lack of model description and presentation of main equations used by ANSYS Fluent. Along with this, the reviewer also identified the lack of citations supporting the choice of ANSYS Fluent (e.g. validation studies).
2. The lack of a salinity and pressure-dependent thermal boundary condition for the ice boundary.
3. Overall issues with clarity in describing model equations and model initialization.

To address these concerns, we have revised our appendix to present the full model framework, including governing equations, and domain and meshing. We have referenced validation studies in the main text. In addition, we have re-run each of our simulations with a salinity and pressure-dependent ice thermal boundary condition.

We address each comment below. The reviewer's comments are written in black, our responses are given in red, and the original manuscript text is in grey.

First Comment:

The governing equations that the code solves should be clearly presented, as I do not think that TC readers are familiar with RANS models (and because I have never seen RANS models used in this context). This requires presenting the Reynolds decomposition of the flow (which would help you justify the fact that 2D dynamics is expected since turbulence is not resolved) and the governing equations for the ensemble-average variables. The closure for the Reynolds stresses and turbulent fluxes should then be discussed in greater details than in the current manuscript. The kappa-eps scheme is mentioned (Eq. (2)) but important details are lacking: for instance, how are kappa and epsilon related to the resolved variables?

We have added a section to the appendix describing the governing equations of the model and further discussion on the turbulence closure scheme. This includes the RANS equations, a description of the Reynolds stresses and closure problem, as well as the choice of turbulence closure used here. Within this section, we have also described the choice to modulate C_μ, as brought up by Reviewer 2, and explained what such modulation does to the flow regime.

I would like also to see more details on the so-called damping functions of the low-Re formulation that supposedly enable accurate diffusive boundary layer representation. Is it like in a wall-resolved large-eddy simulation model?

The standard k-e turbulence model relates eddy viscosity to turbulent kinetic energy and turbulent dissipation to close the system of equations. This requires solving the transport equations for turbulent kinetic energy and dissipation. Near the wall, turbulent kinetic energy vanishes, which produces a singularity in these equations. In order to fix this, a modification must be made to the timescale set by the ratio of turbulent kinetic energy and dissipation. This is where low-Reynolds formulations of the k-e model are helpful, which resolves the transport of turbulent kinetic energy and turbulent dissipation in the low-Reynolds number regions of flow (i.e near no-slip boundaries).

Other formulations of low-Reynolds models are also available within ANSYS Fluent, and we have conducted a sensitivity analysis to determine the most appropriate version. Within the appendix, we have included an explanation of the exact version we use, alongside a description of how it works. We have edited parts of the appendix as shown in the in-line comment #6 below.

Second Comment:

The decoupling of the salt dynamics from the ice-ocean boundary dynamics is not justified and a priori seems wrong. I assume that this decoupling is due to code limitations. However, if you cannot justify the decoupling, I am afraid this just means that the code is not suited for environmental flows with temperature and salt stratification and melting. In the worst case scenario you might consider reformulating the problem in terms of a single scalar variable, namely thermal driving (ref: Adrian Jenkins' papers and other people's related works). The collapse of the full dynamics onto a reduced thermal driving model is thought to be accurate in highly-turbulent environments (for which kappa-epsilon applies anyway) and small salinity variations. This latter condition is obviously problematic (which should be discussed) with regards to your problem of interest. However, I would rather see thermal driving model simulations transparently solved by ANSYS than results from a non-transparant full temperature-salinity model.

Salt dynamics within the fluid regime are resolved and salt is transported as a 'tracer' with an advection-diffusion equation:

$$\frac{\partial}{\partial t}(\rho S) + \nabla \cdot (\rho \vec{u} S) = -\nabla \cdot \vec{J} + R_i$$

Where ρ is fluid density, $\vec{u}$ is the velocity vector, $\vec{J}$ is diffusion flux, and R_i is the net production rate.

The only way salt is not considered is within the melting dynamics. This is in two ways, first, we do not resolve melting or freezing from dissolution – where salt would get into the crystal matrix, suppress the freezing point to below the ice temperature, and therefore cause it to melt. This process would only dominate ice loss if the seawater is subfreezing and therefore mass transfer into the ice matrix would exceed heat transfer.

The other way salinity could affect melting dynamics is by suppressing the ice's thermal boundary condition. In the original simulations, we set the boundary temperature to $0° \ C$. However, based on suggestions from both reviewers, we have re-configured the simulations to have this pressure and salinity-dependent thermal boundary condition for the ice boundaries.

Third Comment

The boundary conditions, especially at the ice-ocean interface, should be clearly presented and discussed.

We have further elaborated on the boundary conditions (thermal and kinematic) in both the main text and the appendix where the model formulation is discussed.

The physical motivation for the so-called melt-activated formulation is lacking. Melting produces buoyant flows even when the boundary is no slip, simply because melting acts like a sink for salinity (though this is lacking in your model).

Since salt transport is resolved in the model (i.e. salt is transported as a tracer) there does not need to be a sink for salinity in the melting framework. The model solves for the displacement and movement of 'saltier' waters via the salt transport equations. Mass is conserved via a pressure outlet (zero gradient flux boundary condition) at the top of the 'ocean' domain.

Thus, should we envision your melt-activated formulation like a velocity compensation for the lack of salt sink in the model? If so, it was not clear to me whether the velocity is prescribed vertically or horizontally, and I am not sure it is the correct way to compensate the salt sink.

The velocity is not compensating for the salt sink, it is the rate at which buoyant freshwater is input into the domain to mimic melting. The velocity is prescribed normal to the boundary where it is sourced, in this case vertical.

Estimating the velocity to enforce at the boundary to mimick the salinity sink from Eq. (4) is also not justified, i.e. why should the movement of the interface (assuming there is no immediate hydrostatic equilibrium of the ice shelf at such small scales) be directly used as a buoyancy-driven velocity input?

We acknowledge that adding a vertical velocity to mimic the vertical movement of a solid is a bit crude, however, because this vertical velocity (the melt rate) is small relative to the main flow and is negligible in the momentum equations, we believe the decision is justified.

Fourth Comment

In would like to see a validation of the code, which includes the choice of turbulence closure. A simple benchmark case should be set-up, for which turbulence-resolving simulation data exist (either from direct numerical simulation (DNS) or large-eddy simulation (LES)). Several groups (with Catherine Vreugdenhil, John Taylor, Ken Zhao etc) have published such DNS and/or LES data over the past 5 years or so, making it practical. Typical configurations are channel flow configurations, which should be accessible to ANSYS. Validation could be based on quantitative comparisons of mean variable profiles (e.g. temperature, TKE) normal to the ice-ocean interface.

ANSYS Fluent was chosen to address this research question because of its extensive validation and history of practical use within the engineering community. Examples of validation exist for many different flow regimes, here are a few about environmental flows:

- Chalá, D.C.; Castro-Faccetti, C.; Quiñones-Bolaños, E.; Mehrvar, M. Salinity Intrusion Modeling Using Boundary Conditions on a Laboratory Setup: Experimental Analysis and CFD Simulations. *Water* **2024**, *16*, 1970. https://doi.org/10.3390/w16141970

- Zangiabadi, E.; Edmunds, M.; Fairley, I.A.; Togneri, M.; Williams, A.J.; Masters, I.; Croft, N. Computational Fluid Dynamics and Visualisation of Coastal Flows in Tidal Channels Supporting Ocean Energy Development. *Energies* **2015**, *8*, 5997-6012. https://doi.org/10.3390/en8065997

- Chan, S.N., Lai, A.C.H., Law, A.W.K., Eric Adams, E. (2020). Two-Phase CFD Modeling of Sediment Plumes for Dredge Disposal in Stagnant Water. In: Nguyen, K., Guillou, S., Gourbesville, P., Thiébot, J. (eds) Estuaries and Coastal Zones in Times of Global Change. Springer Water. Springer, Singapore. https://doi.org/10.1007/978-981-15-2081-5_24

- Rana A. Al-Zubaidy, Ali N. Hilo, (2022). Numerical investigation of flow behavior at the lateral intake using Computational Fluid Dynamics (CFD). Materials Today: Proceedings. Volume 56, Part 4. Pages 1914-1926. ISSN 2214-7853. https://doi.org/10.1016/j.matpr.2021.11.172.

- Sultan, R. A., Rahman, M. A., Rushd, S., Zendehboudi, S., & Kelessidis, V. C. (2018). Validation of CFD model of multiphase flow through pipeline and annular geometries. *Particulate Science and Technology*, *37*(6), 685–697. https://doi.org/10.1080/02726351.2018.1435594

- Oh, CH, & Kim, ES. "Validations of CFD Code for Density-Gradient Driven Air Ingress Stratified Flow." *Proceedings of the 18th International Conference on Nuclear Engineering. 18th International Conference on Nuclear Engineering: Volume 6*. Xi'an, China. May 17–21, 2010. pp. 201-209. ASME. https://doi.org/10.1115/ICONE18-29807

We believe any additional validation would be outside the scope of this paper.

Fifth Comment

Successful applications of ANSYS Fluent RANS solver to environmental flow configurations with temperature and salinity stratification should be cited and discussed (in particular how they validated the code).

We appreciate the reviewer highlighting the lack of justification for choosing ANSYS Fluent within the paper. We have incorporated examples of validation and applications of ANSYS Fluent to stratified and/or complex environmental flow scenarios.

Sixth Comment

The discussion of the steady state and transient runs is really confusing. You should distinguish the existence (or non-existence) of a steady state from your strategy of successive runs to achieve it. The key point that should be in the main text is that the problem has a natural steady state (for the ensemble-average variables) as there is no external variability and the ensemble-average variables do not exhibit temporal fluctuations once equilibrated. The strategy to reach it should then be discussed in an appendix.

We have revised our methodology to determine if a steady state has been reached. Instead of switching between solver types (transient vs. steady-state), we use only the transient solver. The domain is initialized with a salty, warm ocean tank and fresh, cold subglacial environment (e.g. Figure B1 in the revised draft). The simulation then runs for 8640 time steps with a time step of 5s. A quasi steady-state for most simulations is reached during this time, as decided by a change of less than 0.0001 kg/m^3 over several timesteps in the spatially averaged subglacial environment density. The simulations that did not reach a quasi-steady state in the first 8640 time steps were run for another 8640 time steps. The results presented in the main text are time-averaged from once a quasi-steady-state is reached. We have elaborated on this methodology in the methods section and the appendix.

Seventh Comment

I have found many typos in the appendices ("Need to list reference values, materials info, the methods and controls" found line 566), suggesting that these were not carefully reviewed by the authors. At the moment the appendices seem primarily like a draft list of comments and figures that did not make it into the main text (some in fact repeating what is already in the main text).

A careful review of the appendix has been conducted. In addition, we have added further descriptions of the governing equations (major comment 1), model boundary conditions (major comment 3), designating a quasi-steady-state, and analysis of turbulence parameters. In addition, we have moved extra simulations and figures to the supplementary document to make the appendix more concise.

Eighth Comment

The figures are lisible but could be improved(e.g. the x-axis label of Fig. 6 is quite small).

We have standardized the figures' text sizes and shapes for all figure axes and titles.

Ninth Comment

Because of the many questions I had/have with respect to the simulation code, I was not able to appreciate the comparison of the simulation results with the parameterized predictions of melt rates. If the authors can validate their code, I agree that this comparison would be an important addition to the paper, but it would have to be a fair comparison. That is, if the authors end up solving a model that is distinct from the model that the parameterizations (necessarily approximate) aim to mimic (arguably the real exact model), they should discuss result differences in light of model differences.

We understand the concerns of the simulation code to be: salinity sinks, ice-ocean boundary temperature independence of salinity, and vertical velocity as a proxy for a moving ice interface.

We reiterate that salt transport is resolved in these simulations and the only way it is not represented in the melting framework is by having a suppressed boundary temperature. Including this latter point would only increase our simulated melt rates, by increasing the thermal driving. Since the parameterized melt rates are already lower than the simulated melt rates, including a boundary temperature dependent on near-ice salinity would further increase the disagreement between simulated melt rates and parameterized melt rates. Our discussion of the simulation-parameterization disagreement attempts to highlight the differences in light of the assumptions of the fluid structure that are inherent in the parameterization. i.e. well-mixed, higher Reynolds number, etc.

However, considering comments from both reviewers, we have re-run each simulation with a thermal boundary condition that is dependent on near-wall salinity and a reference glaciostatic pressure (i.e. at 1000 m).

Since salt transport is represented in the model by its advection-diffusion equation, a sink does not need to be provided where meltwater enters the domain. Mass is conserved via a pressure outlet (zero gradient flux) downstream in the domain.

We acknowledge inputting a vertical velocity as a proxy for a moving ice interface is crude, however, because this vertical velocity is O(-6) and the freestream flow is O(-1) – O(-3) we think this choice is justifiable.

Tenth Comment

Simulation snapshots and movies would really help visualize the flow.

We have included a figure representing the initial state of the flow in the revised appendix (Figure B1).

Additional comments, including technical corrections: typing errors, etc.

1. line 29: could you clarify the idea of "tidally asymmetric"?
    - Added "Such asymmetry results in stronger melting during the ascent of high tide and weaker melting during the tidal ebb." To the next line.
2. line 87: writing "The ice wall boundaries have a temperature boundary condition of 0◦C" really felt like a bomb! And the lack of justification or description of the full mathematical model did not help disarm it. It is really not expected that prescribing 0◦C at the ice-water interface is reasonable, especially when the salinity goes from 0 to 30 psu.
    - Based on suggestions from both reviewers, we have revised this thermal boundary condition to be salinity and pressure dependent following :

      $$T_b = S\lambda_1 + \lambda_2 + z\,\lambda_3$$

      We have set the pressure dependence to a reference state of 1000m thick ice column.

3. line 108: this equation of state may be suited for small salinity variations, but with a range of 0 to 30 psu, it may not be appropriate, unless the grounding line is beneath a lot of ice. I would recommend that you use a higher-order approximation of the true equation of state, or at least acknowledge that you know of the anomaly of the equation of state at low salinity and pressure but that you discard its effects to simplify the problem. Ideally you could discuss/speculate on how the results might change should you consider an accurate equation of state.
    - We appreciate you bringing this concern to our attention. We have run a sensitivity test comparing our linear E.O.S. to the higher-order approximation in Roquet et al. (2015):
        1. $\rho' = -\frac{C_b}{2}(\Theta - \Theta_0)^2 - T_h Z\Theta + b_0 S_A$
        2. $C_b = 0.011\ kg/m^3K^2$
        3. $T_h = 2.5\ x\ 10^{-5} kg/m^4K$
        4. $b_0 = 0.77\ kg/m^3(kg/g)$
        5. $\Theta_0 = \ -4.5°C$

      The results of this sensitivity test are discussed in the revised appendix.

4. line 113: conduction, diffusion, and molecular dissipation all sound like the same thing to me. Could you explain how they differ? (reading your appendices it looks like conduction might be turbulent conduction)

- Conduction represents the transfer of heat due to the thermal gradient. Within ANSYS Fluent, the conductive value used is the 'effective conductivity' which is the summation of the thermal conductivity of the fluid and turbulent conductivity. The diffusive term represents heat transfer due to species diffusion – in this case, salt. In equation A7, this term is the sum of every species' diffusive flux multiplied by their enthalpy. Molecular dissipation is another way of phrasing viscous dissipation, which is the heat transfer due to viscous forces, represented by the effective shear stress and kinematic viscosity.
- Edited line 113 : Energy, and therefore fluid temperature, is evolved via an energy conservation equation employed by the CFD solver resolving advection, conduction, salt diffusion, and viscous dissipation ~~molecular dissipation~~. (ANSYS Inc., 2024). Conduction represents heat transfer due to thermal gradients, and viscous dissipation is the transformation of kinetic energy into thermal energy due to shear forces. As salt diffuses in the medium, it also transfers heat due to its unique thermal properties, and therefore must also be included.

5. line 124: the quadratic quantity indicated is one among many, such that the sentence does not read well. Please reformulate.
    - Edited lines 123 - 127 shown below.
6. line 126: could you recall what the Boussinesq hypothesis is?
    - Edited lines 123-127: A closure scheme is necessary because averaging the RANS equations introduces Reynolds stresses due to turbulent motion within the fluid. Reynold's stresses take the form $\overline{u_i' u_j'}$, the averaged product of turbulent velocity fluctuations. One class of closure models employs the turbulent-viscosity hypothesis, which relates the deviatoric Reynolds Stresses to the mean strain rate via a positive scalar eddy viscosity (Pope, 2000). Here, we utilize the two-equation $k - \epsilon$ closure scheme, which solves for the eddy viscosity by:

$$\mu_T = \rho_w C_\mu \, k^2/\epsilon$$

    - Added to the appendix: The turbulent-viscosity hypothesis (also known as the Boussinesq Hypothesis) is used in turbulence modeling to solve for the Reynolds stresses. This hypothesis states the deviatoric Reynolds stresses (those deviating from the mean) are proportional to the mean strain rate tensor by a positive scalar. This scalar represents the eddy viscosity (also referred to as turbulent viscosity). This relationship is:

$$-\rho\overline{u_i u_j} + \frac{2}{3}\rho k \, \delta_{ij} = \rho\mu_T\left(\frac{\partial \overline{U_i}}{\partial x_j} + \frac{\partial \overline{U_j}}{\partial x_i}\right)$$

The only unknown left in the system of equations is the eddy viscosity, which can be solved for by a variety of different turbulence closure schemes. Here, we employ the two-equation $k - \epsilon$ closure scheme which solves for eddy viscosity by relating it to the square of turbulent kinetic energy and inverse of turbulent dissipation by a positive scalar C_μ. This closure scheme requires two additional equations to solve for turbulent kinetic energy and turbulent dissipation. These equations are:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k\, u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b - \rho\epsilon - Y_M + S_k$$

For turbulent kinetic energy and:

$$\frac{\partial}{\partial t}\,(\rho\epsilon) + \frac{\partial}{\partial x_i}(\partial\epsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial \epsilon}{\partial x_j}\right] + C_{1\epsilon}\frac{\epsilon}{k}(G_k + C_{3\epsilon}G_b) - C_{2\epsilon}\rho\frac{\epsilon^2}{k} + S_\epsilon$$

For turbulent dissipation. Note, in the equation for turbulent dissipation, turbulent kinetic energy is in the denominator which results in issues when k approaches zero near wall boundaries. To resolve boundary layer dynamics with the k-e closure, we use the low-Reynolds number formulation which employs damping functions and fixes the singularity that arises with low values of k.

A version of the low-re k-e closure we employ here is the Yang-Shih version (Yang and Shih 1993). In this formulation, the authors set the near-wall turbulence timescale to be the Kolmogorov timescale ($T_k \;\propto\; (\nu/\epsilon)^{1/2}$). In doing so, the equation for eddy viscosity becomes:

$$\nu_T \;=\; C_\mu\, f_\mu\;\, k\;\,\left(T_k + \frac{k}{\epsilon}\right)$$

Where f_μ is the "damping function" and equal to:

$$f_\mu \;=\; \left[1 - exp\left(-a_1\, R_y - a_3\, R_y^3 - a_5\, R_y^5\right)\right]^{1/2}$$

and $R_y = \frac{k^{\frac{1}{2}}y}{\nu}$. The constants a_1, a_3, and a_5 are constrained from DNS experiments for turbulent channel flow.

The final adjustment to the standard k-e formulation for near-wall flows is to add an additional source of dissipation, which results from inhomogeneity in the mean flow field. This takes the form:

$$E = \nu \nu_T \frac{\partial U_i}{\partial U_j \partial U_z} \frac{\partial U_i}{\partial U_j \partial U_z}$$

This formulation of the low-re k-e turbulence closure allows for the free-stream portion of the flow regime as well as the near-wall region where viscous effects dominate since the added terms tend to zero when turbulence is high.

7. Eq. (2) P2: what are the equations for kappa and epsilon? I expect that they involve the resolved ensemble-average variables.
   - We have added these equations to the appendix in the model description as written above in comment 6.
8. Section 2.4: this section is really confusing as I already mentioned earlier, because buoyancy isn't due to some interfacial velocities but to changes in salinity and temperature at the phase-change boundary. Please reformulate.
   - We have reformulated how we discuss the buoyancy effects that arise due to the input of fresh, cold water. We have emphasized that any buoyancy effects are exclusively due to the difference in density between the input meltwater and the ambient water, and not due to some prescribed velocity (which wouldn't be a buoyancy effect).
9. line 150: I don't see the physical justification for prescribing a "horizontal ice (?!) velocity" at the horizontal ice-water interface.
   - A fluid velocity-inlet boundary condition is applied to the horizontal ice base when melting is enabled. The fluid velocity prescribed here is normal to the boundary and therefore is downward, not horizontal.
   - Edited line 150: The downward fluid velocity prescribed at the horizontal ice face is set by the melt rate, $\dot{m}$, and is a function of the difference between the near-wall cel''s centroid temperature T_w and the ice-ocean interfacial temperature T_b, thermal conductivity k_T, and density of ice ρ_i:
10. line 179: can you provide a reference for "realistic estuarine-like mixing rates"?
    - We appreciate the reviewer bringing the lack of citations to our attention. We have amended line 179 to:
      1. Turbulent mixing, as modulated by C_{μ}, affects intrusion distance to a lesser degree than freshwater discharge velocity when varied over a wide range encompassing likely values on the lower-end for realistic estuarine-like mixing rates (Geyer et al. 2000, Geyer et al. 2008).
11. line 193: rewrite "retrograde bed slope".
    - We have done this.

12. You refer to figure 5 before figure 4 so the two should be swapped.
    - We have done this.
13. line 253: Could we say "vertical baroclinic convective motion"?
    - We appreciate this suggestion and have made this edit.
14. Eq. (9) line 325: this equation doesn't look like a parameterization but rather like the exact expression for the melt rate as a function of the conductive heat flux at the interface.
    - Based on both reviewers' suggestions, we have removed this section from the results.
15. Fig. 5-7: it was not clear to me whether the results you plotted were for the melt-enabled model or not.
    - In Figure 5, the dashed lines represent melt-enabled scenarios as denoted in the caption. To address confusion, we have added:
        - When melt is enabled the horizontal extent of stratification in the subglacial environment is reduced, but where stratification occurs, it is stronger (e.g. dashed lines in Figure 5).
    - For Figure 6, all data comes from melt-enabled cases, as denoted in the caption. We have added to line 341 to improve clarity:
        - Here, we tested the sensitivity of equations 6-9 to various choices of ice distance to obtain Tw, Sw, ρw, and u for the melt-enabled cases.
    - For Figure 7, both melt-enabled and no-melting cases are represented as denoted in the legend. We have edited the caption to improve clarity:
        - Wilson et al. (2020) experimental data (gray markers) and intrusion characteristics found in this study (red and blue markers). The red markers represent simulations with melting enabled, and blue markers represent non-melting simulations. The black dashed line is the numerical solution to Robel et al. (2022) with $\gamma = 2$.

    In addition, we have added to lines 387-389:

        - Our simulated intrusions for both non-melting and melt-enabled scenarios follow the general trend and scale sensitivity to those identified in previous laboratory experiments (Figure. 7) (Wilson et al. 2020) which are within a factor of 10 to the theoretical prediction (dashed line) from Robel et al. (2022).

16. Eq. (A3): should it be Re_L? Or change L into x?

    - We appreciate you pointing out this inconsistency. We have amended the equation to be:

        - $Re_L = \frac{uL\rho_w}{\mu}$

17. Appendix A4: I have found many typos, you need to use capital letters for Reynolds number, kappa has become k etc Eq (A7) has signs/symbols displaced. Please discuss Eq (A7) more carefully. What is tau_eff? What is species diffusion in your case?

    - In combination with previous comments (e.g. 3, 4,5, 6, 7, and major comments 1 and 5) we have revised the appendix to describe the model in more detail and justify model choices. In this rewrite, we have standardized the symbols used and refined appendix equations.
    - T_{eff} is the effective shear stress, which includes viscous shearing effects as well as shear from the no-slip boundary conditions.
    - Species Diffusion in our case represents salt diffusion. Since we are transporting salt as a tracer throughout the fluid, we must include its heat transport into the energy equation.

Works Cited

Pope, S.B. (2000). Turbulent Flows. *Cambridge University Press*, Cambridge, 305-308.

Geyer, W.R., Trowbridge, J. H., & Bowen, M.M. (2000) The Dynamics of a Partially Mixed Estuary. *J. Phys. Oceanogr.* **30**, Iss 8, 2035-2048. https://doi.org/10.1175/1520-0485(2000)030<2035:TDOAPM>2.0.CO;2

Geyer, W.R., Scully, M.E. & Ralston, D.K. (2008). Quantifying vertical mixing in estuaries. *Environ Fluid Mech* **8**, 495–509. https://doi.org/10.1007/s10652-008-9107-2

Roquet, F., Madex, G., Brodeau, L., Nycander, J. (2015). Defining a Simplified Yet “Realistic” Equation of State for Seawater. *J. Phys. Oceanogr.* **45**, Iss 10, 2564-2579. https://doi.org/10.1175/JPO-D-15-0080.1

Response to Reviewer #2 for Manuscript "High-Fidelity Modeling of Turbulent Mixing and Basal Melting in Seawater Intrusion Under Grounded Ice" by Mamer, Robel, Lai, Wilson, and Washam

Summary:

This paper studies the estuarine-type dynamics of seawater intrusion/subglacial discharge upstream of an ice sheet grounding line in a CFD model (ANSYS fluent). The subglacial environment is two-dimensional with a large aspect ratio (5cm high and 3-5m long) and is forced by fresh subglacial discharge upstream and salty inflow from the ocean downstream. The authors investigate the structure and distance of the saltwater intrusion, and its sensitivity to various parameters (primarily freshwater discharge velocity). They find that that the height of the subglacial environment and the velocity of the freshwater discharge are strong controls on the intrusion distance, and the strength of turbulent mixing has a somewhat weaker/more ambiguous effect. The authors also investigate basal melt at the upper surface of the subglacial region and how it affects the intrusion (to do so they must change the boundary condition at the top surface away from the no-slip condition), finding that melt decreases the intrusion distance, offering a possible negative feedback.

This is an interesting study of an important process; however, I have several major concerns with the paper that may affect the conclusions. If they are addressed, I would be happy to review a revised manuscript.

We appreciate the thoughtful review and valuable feedback given by reviewer 2. We have identified the reviewer's main concerns to be:

1. The lack of turbulence statistics in describing differences between laminar and turbulent cases and in justifying the modeling choices to vary C_μ.
2. The lack of a pressure and salinity dependent thermal boundary condition for the ice boundary. Furthermore, the lack of a free-slip case to compare the melt-enabled simulations to.
3. The incomplete discussion of a double-diffusive/diffusive mixing regime and inappropriate diffusive melting comparison to modeled melt.
4. Overall issues with clarity in describing the numerous simulations.

To address these concerns, we have re-run each of our simulations with a salinity and pressure-dependent thermal boundary condition and added additional cases with a free-slip ice kinematic boundary condition. We have provided a turbulent viscosity figure in the appendix (Figure D1) to aid in discussing the turbulence metrics. The consideration of diffusive-convective mixing has been limited to the discussion section and removed from the results section. We have provided tables of the various simulations to improve results clarity.

Below, we address each concern in greater detail. The reviewer's comments are written in black, our responses are given in red, and the original manuscript text is in grey.

First Comment:

I am concerned about the turbulence modelling and the lack of inclusion of turbulence quantities in the paper. The fidelity (reported to be high!) of these simulations relies on the appropriateness of the turbulence closure, however, the reader is not given any context for the choice/s of C_mu,

The typical value assigned to C_μ is 0.09, which has been empirically found for simple wall-bounded flows (Pope, 2000). However, for complex stratified flows, or highly energetic jets, a standard constant value for the whole domain may not be appropriate (e.g. Lai et al., 2019). Since C_μ could be found via the relationship:

$$C_\mu = \frac{\nu_t \epsilon}{k^2 \rho}$$

it is clear that the 'real' value of C_μ is highly dependent on turbulence dynamics. In a modeling framework, we have to prescribe C_μ in order to obtain eddy viscosity, it is not a post-processing derived value. Since a 'true' or 'real' value for C_μ has not been found for the flow regime considered here (stratified), we deemed it an appropriate approach to modulate this value in order to induce more or less turbulent mixing. Our original intention in setting C_μ was to span three orders magnitude, with the standard value of 0.09 as the 'middle case'. However, initial runs demonstrated instability within the model when C_μ was set to 0.9. On the other end, when C_mu is very low ($\rightarrow$ 0) the flow regime becomes laminar, which would not be helpful to address the research question at hand. Based on this, we chose to range the value by a factor of ¼.

nor are we given any more intuitive quantities (e.g. the resulting turbulent diffusivity) to better understand the effect of varying C_mu, and how increased turbulence affects the flow.

We agree that turbulent viscosity would be more intuitive in explaining what changing the C_μ value results in. We have included a figure in the appendix depicting turbulent viscosity values across all freshwater velocities and turbulence levels for the standard geometry (Figure D1).

Without this information and context, I have low confidence in the modelling overall, especially since the effect of varying C_mu is non-monotonic for some cases (Fig 2B) and counter to my expectations for others (Figs 2A & C), i.e. I would have expected that increased mixing would decrease intrusion distance, whereas the authors find the opposite result. My recommendations to address this are as follows:

- Present the turbulent diffusivity (or viscosity) alongside T, S, u. In main MS or in a supplement

This has been done and is appendix figure D1 in the revised manuscript.

- Present the Reynolds numbers for the cases

This has been done. The Reynolds numbers are approximately 25, 250, and 2500 for the low, medium, and fastest freshwater cases presented.

- Give some context for the choice of C_mu for similar flows – I presume the engineering literature can provide. Stratified plane couette flow may be a starting point in terms of a bounded, stratified flow.

To increase turbulent mixing, we could either increase the flow velocity or increase the eddy viscosity via C_μ. Since we want to look at low Reynolds numbers, we modify the C_μ value to mimic a changing turbulent mixing scenario.

In setting C_μ, we found numerical instability when $C_\mu \to 1$, therefore limiting our upper-end choice to a factor of 2 of the standard value used. For the lower-end case, we wanted to avoid choosing a too small value, since the solution relaxes to laminar flow when $C_\mu \to 0$. This constrained our lower-end choice to a factor of 0.5 of the standard value.

The empirically derived value for C_μ that is traditionally used, 0.09, is for standard wall-bounded simple flow regimes. Under scenarios of high energetics (like a jet-plume) the C_μ value can approach 0.3 (Lai et al., 2019). In parts of our domain, a jet-plume does form, therefore making a relatively increased C_μ value realistic.

Second Comment:

I don't understand why the temperature profiles have much sharper gradients than the salinity gradients (c.f. the darkest line in Fig A8A to the darkest line in Fig A8B). The authors mention the steep T gradients (not seen in S) in line 202 *(Such a steep thermocline is most likely due to the temperature boundary condition we imposed on the horizontal ice boundary)* however, I am concerned that the problem runs deeper than different boundary conditions. Why, for example, is temperature at 2cm depth at the GL entrance at 0.5 degrees (i.e. unmodified from ambient conditions), while salinity is <20ppt at the same location? There may be a serious issue with the mixing of these scalars which would significantly affect your results. Temperature-salinity plots may help determine if there is a problem.

We appreciate the reviewer bringing this to our attention. We have found a mis-assigned diffusion coefficient. We have re-run all simulations with this fixed diffusion coefficient and it has increased intrusion distances for all cases (Figure 2 in the revised manuscript). Furthermore, the distribution of heat and salt are now more realistic (Figure 3 in the revised manuscript). We have also provided TS diagrams in the supplementary for both melt-enabled and non-melt enabled simulations for all geometries and freshwater velocities tested.

Third Comment

By comparing the temperature and salinity profiles as u_f is increased (Figs 3 A,B, Figs A8 A,B,D,E), it appears to me that the greatest control on mixing is time spent in the subglacial channel. For example, at u_f=0.05 cm/s salinity is quite well-mixed over the full depth of the channel, and varies mostly with distance along-channel, implying that diffusion (rather than advection) is dominating transport. For u_f=5cm/s, the top, outflowing layer remains fresh, indicating that the transport dominated by advection. This is somewhat counter to my expectations that turbulent mixing should be stronger for the higher velocity cases. This result should be investigated and discussed. As for comment (1), plots of the turbulent diffusivities for each case may be enlightening.

We have revised Figure 3 to include vertical profiles of thermal forcing, salinity, velocity, and buoyancy frequency across all freshwater velocities tested. This updated figure illustrates the relationship described above: slower freshwater velocities exhibit more uniformity in the vertical direction, while faster velocities lead to higher stratification. This pattern is similar to what is observed in estuaries and is likely due to boundary shear being a more dominant factor in slower freshwater flows. In estuaries, river outflow must overcome tidal shear and wind-driven surface shear to maintain stratification. Similarly, under ice, the fluid must overcome shear from both the ice and rock boundaries.

Fourth Comment

The different BC between the melt-enabled and no-melt cases makes it impossible to attribute changes in the intrusion to the effect of melting alone. An additional case is needed with free slip and no melting to better separate these effects.

We have conducted additional free-slip cases to compare the melt-enabled cases to. These results are presented in Figure 2 and Table 2 in the revised manuscript.

Fifth Comment

There are numerous different simulations included in the paper, however, few where one variable is systematically changed. This makes the paper/results hard to follow at times. A results table or bar chart showing how the intrusion distance changes across simulations would go some way to addressing this.

We have added three tables in the main text (Tables 1-3) and 2 tables to the appendix (Tables B1 and B2) to describe the model setup, parameters, and intrusion distance. We did not include the drag coefficient and melt rate since they vary significantly over the length of the intrusion.

Sixth Comment

No salinity effect on the melt. In the simulations, the interface is salty (Fig 3E) which will act to depress the freezing temperature and therefore the interface temperature, altering the melt rate. The authors should state why they have not included this extremely important effect (in more detail than "model limitations" line 437). In addition, some discussion of the likely effect of this simplification on the results is needed.

We have re-run all simulations with an updated thermal boundary condition on the ice face following the salinity and pressure dependent freezing point:

$$T_b = S\,\lambda_1 + \lambda_2 + z\lambda_3$$

We have set z to be 1000 m and also prescribed the inflowing freshwater boundary to be at the pressure dependent freezing point.

Seventh Comment

Double diffusive framework. Equation 9 is the same as Equation 4 (with marginally different thermal diffusivity) and is therefore not a parameterization to be tested, rather a re-stating of your melt BC. I propose removing Fig 6B and the "diffusive melt" in Fig 5. Notation m_DC and m_DDC (which are interchangeably used) are inappropriate here.

We have removed the double diffusion framework from the results section, limiting the parameterizations to the heat-limited two-equation formulation of Holland and Jenkins (1999). However, we have kept the consideration of the double-diffusive melting mechanism to the regime of seawater intrusion within the discussion section.

Eighth Comment

Transfer velocities/Stanton numbers (Figure 6)– I don't really understand the purpose of C and D. All D shows is the weak dependence of the Stanton number on ustar at low ustar (see the denominator of (7)), and all C shows is the same but multiplied by ustar. Since the S99 parameterization can't accurately predict melting for your simulations, this (trivial) result becomes misleading, as readers may think that it lends support to a certain Stanton number being useful more broadly for modelling ice-ocean interactions. If you want to compare the Stanton number of your simulations to those found in the literature, you need to rearrange equation (6) less the conductive ice flux to solve for the Stanton number (and the (unitless) transfer coefficient \$Gamma_T$) using your model output melt rate, temperature etc.

We have removed this figure and discussion of it from the results section. Instead, we have focused on two ways of using the heat-limited shear driven parameterization framework. In the first approach, we directly calculate the turbulent transfer coefficient along the horizontal ice boundary using the time-averaged simulation data. The second method involves finding a "tuned" turbulent transfer coefficient. We do this by evaluating the melt rate over a range of turbulent transfer coefficients and find which one has the lowest root mean squared error with the simulated melt rate. Both of these are presented in Figure 5 of the revised manuscript and discussed in the results section.

Other Comments

1. Line 83 – move references (carter…) after "non-summer months"
   - Fixed

2. Line 97 - "Our vertical domain size is at the upper bound of the viscous sublayer length scale that could exist between a well-mixed boundary layer and the ice" I don't know what this means, please clarify.
   - The domain size and velocities (Reynolds number) used in these simulations constrain the development of the boundary layer and freestream flow. In this Reynolds number regime, we don't see the development of a complete boundary layer with flow unaffected by the boundary (freestream flow)
   - To clarify, we have made these edits in the main text:
   - ~~Our vertical domain size is at the upper bound of the viscous sublayer length scale that could exist between a well-mixed boundary layer and the ice (i.e. the vertical domain is small and does not include the turbulent outer layer).~~ For the tested freshwater velocities, the vertical domain size hinders the development of a full boundary layer. Instead, everywhere in the domain, the fluid feels the effects of the wall boundary.

3. Line 154 – should be dT/dy in your coordinates
   - Fixed

4. Line 177 – "vary by a factor of 1000 **in response to the range of input velocities tested here**".
   - Fixed

5. Line 179 – insert comma before "increased"
   - Fixed

6. Line 180 – "For the middle freshwater velocity (Figure 2 C)," - should this be 2A?
   - Fixed. We appreciate you finding this error.

7. Line 182 – "To contrast the effects of turbulent mixing, we tested a laminar flow case with no turbulent mixing (green line Figure 2 A) and saw no meaningful difference in intrusion distance." It would be good to compare and contrast this case more, i.e.

how different is the T/S structure? If it's not different, then presumably turbulent mixing is not occurring in the channel.

- We appreciate this suggestion and have included a supplementary figure with vertical profiles for the standard geometry with u_f = 0.5 cm/s across all turbulence levels to compare the effect of turbulent mixing more.

8. Line 197/8 – refs should not be in parentheses.
   - Fixed.

9. Line 212 – How is Cd calculated? i.e. at what value of y are ustar and u evaluated? Also, as mentioned earlier I think Cd is a main result and this figure should come to the main text.
   - To calculate C_d, we use the relationships between drag coefficient and wall shear stress, τ_w (Pope, 2000):
     - $C_d = \tau_w \; / \; (0.5 \; \rho \; \bar{u}^2)$
     - $\tau_w \; = \mu \left(\frac{\partial \bar{u}}{\partial y} \right)$
   - Where ρ is the fluid density, and $\bar{u}^2$, is the mean freestream flow.
   - We can rearrange to solve for C_d:
     - $C_d = \frac{\nu}{2\bar{u}^2} \left(\frac{\partial \bar{u}}{\partial y} \right)$
   - Where ν is the kinematic viscosity and equivalent to μ/ρ, the dynamic viscosity divided by density.
   - The gradient in mean streamwise velocity $\left(\frac{\partial \bar{u}}{\partial y} \right)$ is found by fitting a line to the upper half of the freshwater layer. We provide more details in the revised appendix and supplementary document.
   - We have refined the methodology of calculating drag and put this figure into the main text as Figure 4. The methodology is discussed in various levels of detail in the revised main text, appendix, and supplementary.

10. Line 241 – "Reduction in velocity gradients arises from an increase in stratification, suppressing turbulence, and the kinematic boundary condition being a velocity inlet and not a no-slip wall." – as per major comment 4, actually you can't isolate these effects currently and a no-slip no-melt case is needed.
    - We have conducted free slip cases for all simulation geometries and freshwater velocities to compare the melt-enabled cases to. The comparison between these cases have replace the results quoted above.

11. Line 256 –Predominantly horizontal motion?
    - The motion is predominately horizontal from how the kinematic boundaries are prescribed. However, the horizontal density gradient that arises due to the characteristic 'wedge' shape of the seawater intrusion could introduce vertical convective motion to 'flatten' out the intrusion. This vertical motion

may drive interfacial mixing and is an important mechanism to reduce the strength of stratification.

- To further clarify, we have edited lines 252-258:

  However, the horizontal density gradient introduced by the characteristic wedge shape of seawater intrusion will drive vertical baroclinic convective motion to flatten isopycnals. Such baroclinic adjustment may be an important source of interfacial mixing, working in tandem with turbulence and double-diffusive convection to reduce stratification within the subglacial environment. This convective-driven mixing mechanism differs from convective mixing caused by a sloping ice boundary, in which a buoyant plume may form. For the idealized scenarios in this study, buoyant convection via ice geometry will not drive mixing and thus melt since the ice is perfectly horizontal.

12. Line 261 – again, you can't attribute this to increased stratification (which MAY decrease shear/drag) because you haven't isolated the effect of the no-slip BC (which WILL decrease shear/drag)
    - We have amended any discussion comparing the melt-enabled cases to the non-melting cases to be for those with a free-slip boundary condition.

13. Line 286 – Diffusive convective melting also involves convection driven by cooling and is different from diffusive melting. See Martin & Kauffman (1977) section 3 for diffusive melting and Martin & Kauffman (1977) section 4 for diffusive-convective melting.
    - We appreciate this insight. We have removed the discussion of diffusive convective and double diffusive convective-driven melting from the results section.

14. Fig 3 – Is any averaging (time or space) done to obtain these profiles?
    - Yes, these are vertical profiles from the time-averaged domain. In order to clarify, we have edited the caption:

      Figure 3. Time-averaged vertical profiles of temperature(A, D), salinity(B, E), and x-component of velocity (C, F) along the seawater intrusion for u_f =0.5 cm/s and medium turbulence C_μ = 0.09. The distance beyond the grounding line represents the distance (m) upstream of the fixed grounding line. The top row (A, B, C) is for the non-melt enabled case, i.e. the horizontal ice boundary is a wall boundary with a fixed temperature. The bottom row (D, E, F) is for the melt-enabled case where the ice boundary becomes a velocity inlet with freshwater inflow as a function of near-wall temperature.

      And figure discussion at line 200:

The time-averaged vertical profiles of temperature, salinity, and velocity along the intrusion for non-melt-enabled cases (Figure 3) depict a two-layered flow in opposing directions, with a relatively uniform low-sloping vertical gradient in salinity, and a strong thermocline in the 2 cm directly below the ice.

15. Fig 3 - It would be great to see the systematic change in the intrusion as u_f is increased with a series of side-by-side plots, rather than having to move back and forward from figs 3 to A8.
    - We have presented the vertical profiles for thermal forcing, salinity, velocity, and buoyancy frequency in the revised main text as Figure 3.

16. Line 319 – I would not say that the flow is weak at 5,10 cm/s. However, the height of the channel is very small, so the Reynolds number will be small. Again, Re or other turbulence metrics are needed.
    - All cases tested here have low Reynolds numbers ranging from 24 to 2400. We have included this in the discussion of the methods. In addition, the earlier suggestion to discuss turbulent viscosity metrics has been illuminating and we have used this variable to describe the energetics within the flow field.

17. Line 397 – how is the reduced gravity calculated here? Based on the density difference between the freshwater and saltwater?
    - The reduced gravity is calculated based on the density difference between pure freshwater and pure saltwater. However, a point can be made that the reduced gravity should change along the intrusion and amongst the freshwater cases based on the observation made in major comment 3.
    - To improve clarity, we have edited line 398:
        - Calculating the drag coefficient using model output gives Cd with values of order 10−2 to 100. The analytical theory of intrusion distance (L) for an unobstructed water sheet from Robel et al. (2022) is,

          $$L = \frac{H^2 g'}{4C_d^2 u_f^2}$$

          where H = 0.05 m is the height of the subglacial environment, g' = 0.20 m/s2 is, and Cd is set to the maximum value within the intrusion. Reduced gravity is referenced to the density difference between the prescribed pure freshwater and pure seawater.

18. Line 427 – "If we anticipate viscous effects to dominate in seawater intrusions under grounded ice, then using the thermal and haline molecular diffusivities as so-called "transport velocities" would be appropriate, similar to the diffusive-convective

framework presented above" – The units (m^2/s vs m/s) are not consistent. To turn the molecular diffusivity into a transport velocity, you need a lengthscale, i.e. the width of the diffusive sublayer. That's the hard part which is not addressed here!

- We appreciate the reviewer's insight on this. With the model used here, we have a high enough domain resolution to identify the width of the diffusive sublayer. We can define this width to be where the turbulent viscosity is equal to the molecular viscosity. This works because the diffusive sublayer is where molecular diffusivity dominates. Therefore, there should be a point of transition where turbulent viscosity becomes weaker than molecular viscosity. This transition point (i.e. where they are equivalent) will represent the diffusive sublayer's thickness. We have highlighted this limitation when applying this method to large-scale coupled models.

- We have edited line 427:
  - If we anticipate viscous effects to dominate in seawater intrusions under grounded ice, then using the thermal and haline molecular diffusivities as so-called "transport velocities" would be appropriate. However, this would require knowing the width of the diffusive sublayer which, given computational constraints for coupled ice-ocean models, cannot be resolved.

19. Figure 6 caption – The Stanton number should be γ_T/U, not γ_T/C_d
    - Fixed.

20. Figure 6 caption last line – are both the dashed and two solid lines from Washam et al 2023?
    - Both the dashed and solid lines are reported in Table 1 in Washam et al. 2023. However, the dashed line is reported first in Washam et al. (2020) and is cited as such in Table 1 of Washam et al. (2023). We have ultimately removed this figure from the main text.

21. Line 490 – "bolstering the idea that grounding zones are subglacial estuaries (Horgan et al., 2013).
    - No change is indicated in the comment.

22. Line 566 – seems like a note-to-self.
    - We appreciate you catching this. It has been removed.

23. Line 594 – either "given by (3)" or "given by equation 3"
    - Fixed.

24. The appendix is quite bloated. I think some of the figures could be relegated to SI and some should go to the main text. For example, I think Fig A5 is a key result and should go in the main text. Fig A1 could go in SI.

- Based on recommendations from both reviewers, we have shortened the appendix to contain only descriptions of the model (equations, meshing, run-time settings). We have transferred the discussion of the drag coefficient to the main body under the results section. Sensitivity tests and model evaluation alongside other results have been relegated to a supplementary document.

25. Table A1 – inconsistent unit formatting (italics/roman). Look to TC style guide, or use roman which is typical. Units for theta should just be degrees.
    - We have made all units in Roman format and followed the TC style guide in handling denominator values.

26. Table A2 – again, unit formatting. H [cm]
    - We have made all the units in Roman format, following the TC style guide.

27. Figure A5 – the colours are too hard to tell apart.
    - We have refined this figure and included it in the main text.

28. Figures A9 & A10. The figures are labelled "law of the wall" but no interpretation is offered. What is the black vertical line and what does it mean? What would the profiles be expected to look like if a log layer was present? What portion of the flow is being shown? Is y+=0 at the top or bottom of the domain? Do we see a viscous boundary layer, i.e. u+=y+? in addition, it would be much more helpful if y+ was on the y axis, since that's how the model is set up and how all your other profile plots are oriented.
    - In these figures, the vertical profiles are taken over the top half of the domain (2.5 cm from the ice face), to avoid effects from the intrusion interface. Therefore, y+ = 0 is at the top of the domain, where the ice boundary exists. The black line here represents y+ = 30, which would represent the point at which the log-law region of flow would develop and where u+=y+.
    - For these figures, we have expanded on their interpretation and added a characteristic log law profile. This figure has been relegated to the supplementary.

29. Reference 1 (Adusumilli) seems incomplete
    - Fixed.

Works Cited

Pope, S.B. (2000). Turbulent Flows. *Cambridge University Press*, Cambridge, 305-308.

Lai, C.C.K., Socolofsky, S.A. (2019). Budgets of turbulent kinetic energy, Reynolds stresses, and dissipation in a turbulent round jet discharged into a stagnant ambient. *Environ Fluid Mech* **19**, 349–377. https://doi.org/10.1007/s10652-018-9627-3

Washam, P., Nicholls, K. W., Münchow, A., and Padman, L. (2020). Tidal Modulation of Buoyant Flow and Basal Melt Beneath Petermann Gletscher Ice Shelf, Greenland, J. Geophys. Res.-Oceans, 125, https://doi.org/10.1029/2020JC016427

Washam, P., Lawrence, J. D., Stevens, C. L., Hulbe, C. L., Horgan, H. J., Robinson, N. J., Stewart, C. L., Spears, A., Quartini, E., Hurwitz, B., Meister, M. R., Mullen, A. D., Dichek, D. J., Bryson, F., and Schmidt, B. E. (2023). Direct observations of melting, freezing, and ocean circulation in an ice shelf basal crevasse, Sci. Adv., 9, DOI:10.1126/sciadv.adi7638