RESPONSE TO REVIEWER COMMENTS:

Authors' General comment: Thank you again for the Reviewer's constructive comments concerning our manuscript entitled "Sensitivity analysis of a built environment exposed to debris flow impacts with 3-D numerical simulations" (ID: nhess-2022-173). We have the greatest respect for Reviewer's professional opinions on the debris flow hazard. Reviewer's comments are all valuable and very helpful for revising manuscript and improve our research. We have studied comments carefully and have made corrections which we hope meet with approval. Revised portions are marked in red in the paper and response letter, and the manuscript is re-submitted in clean format to the Journal. Please also find below my response to Reviewer's comments.

REVIEW COMMENTS:

1. The main one concern the validation. In the laboratory experiment used for the validation, a well-mixed stony granular debris flow is reproduced. One of the main characteristics of this debris flow is that the energy dissipation is due to the collision between the particles and not by the viscosity of the fluid (e.g. Iverson 1997, Takahashi 2007, Armanini 2013). However, in the authors' response to my comment #1, it is highlighted that "From the characteristics of RNG k-ε model [that is used in all the manuscript], the type of debris flow involved in this study was determined as mudflow or viscous debris flow, in which a singlephase non-Newtonian fluid was assumed and solid particles were treated as suspension and mixed with the fluid phase well". This statement is completely in contrast with the used laboratory experiment used and consequently, all the section devoted to the validation of the model is meaningless since the author used a model that could not represent correctly the physical processes involved.

AUTHORS RESPONSE: We greatly appreciate for Reviewer's good comments! As Reviewer suggested, we have selected another classic dam-break experiment for the validation of fluid-structure interaction (Gomez-Gesteira and Dalrymple, 2004; Liu et al., 2021). The new statements about the model validation were added in Line 128-157 of the revised manuscript.

Line 128-157:

2.2 Model validation

The interaction between a dam-break and the structure has become a classic benchmark for the validation of fluid-structure interaction (Liu et al., 2021). The accuracy of the model will be validated by means of the experimental setup previously used in Gomez-Gesteira and Dalrymple (2004). This experiment has been referred as a "bore in a box", where it was a dam-break and structure-impact problem confined within a rectangular box. The geometric dimensions of the experimental model are shown in Fig. 1.The rectangular tank is 1.60 m long, 0.61 m wide and 0.75 m high.The volume of water initially contained behind a thin gate at one end of the box is 0.4 m long, 0.61 m wide and 0.3 m high. An initial layer of water (approximately 1 cm deep) existed on the bottom of the tank. The obstacle, which is 0.12 m × 0.12 m × 0.75 m in size, is placed 0.5 m downstream of the gate and 0.24 m from the nearest sidewall of the tank. The time history of the impact force on the structure was measured with a load cell.

In the numerical simulation, the analysis domain was discretized into a grid with a cell size of 0.01 m, which was equal to a cube of 0.01 m side in 3-D model. The fluid properties were set to be the density of 1000 kg m^{-3} and viscosity of 0.001 Pa·s. The motion of fluid was computed by means of RNG k-ε model in FLOW-3D. The obstacle and gate were controlled by the GMO module, specifically this obstacle was set as a fixed and non-deformable rigid body, and the gate was prescribed to be lifted 0.3 m along the Z^+ direction. The time history of impact forces and the corresponding dynamic processes were selected to validate the accuracy of the numerical simulation. The direction of the force was considered positive when exerted in the Y^+ direction.

Fig. 2a shows the agreement of numerical forces obtained by means of the RNG and GMO coupled model with experimental data, particularly the positions of both peaks, which correspond to the wave hitting the front and the back of the structure and were reproduced perfectly by the numerical model. Fig. 2b shows the evolution of the wave generated by the dam-break and the initial layer of water on the bottom. At t = 0.32 s, the wave is colliding with the front of the obstacle. At t = 0.58 s, the wave is wrapping around the structure, colliding together and continues moving toward the tank wall. At t = 1.44 s, the reflected wave is hitting the back of the obstacle.

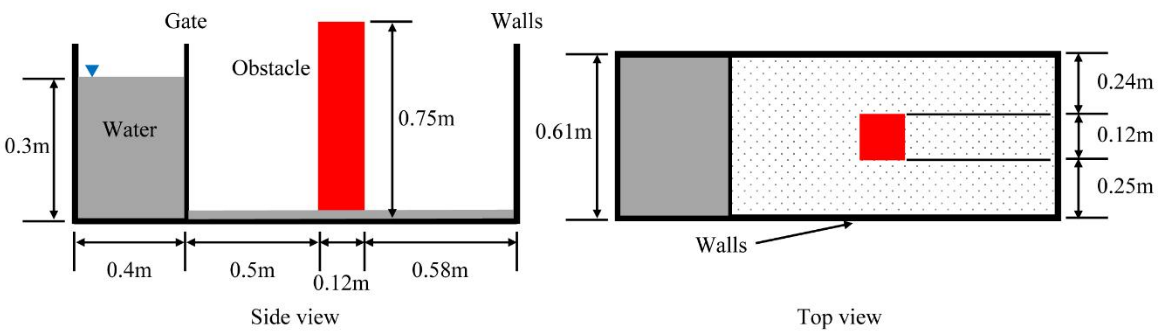

Figure 1. The geometric dimensions of the experimental dam-break model

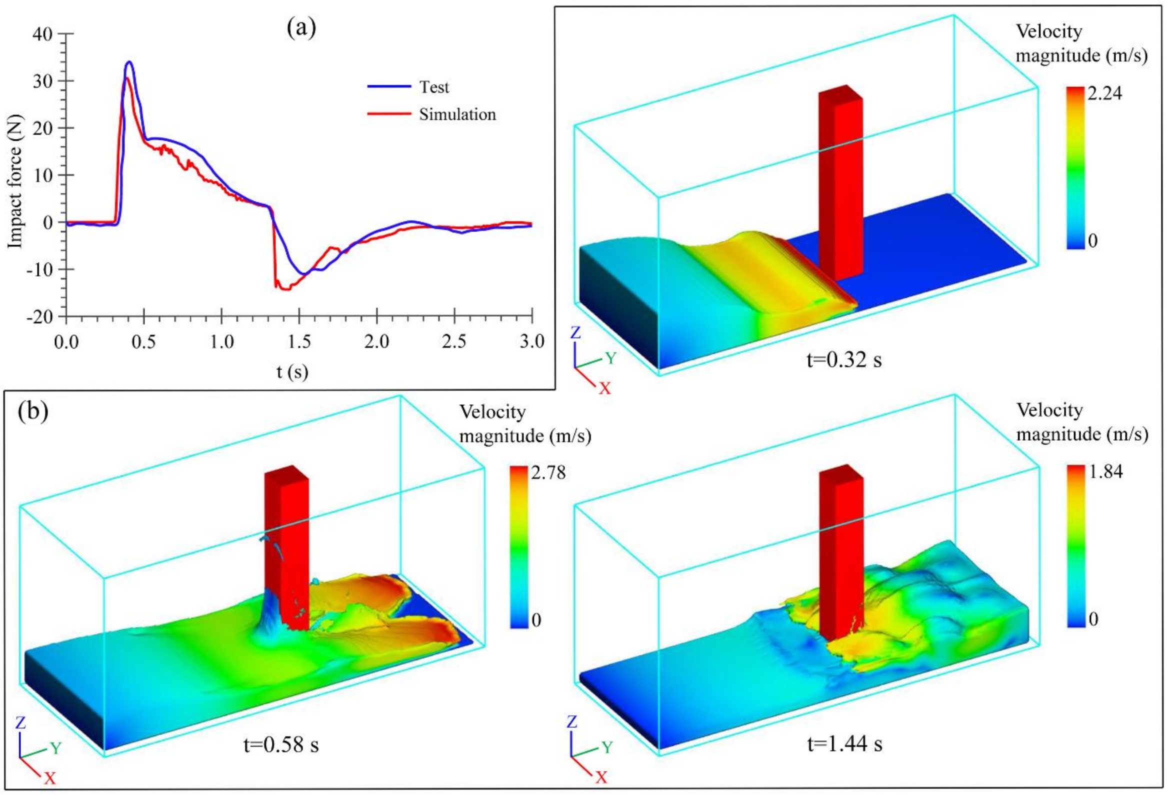

Figure 2. The dam-break simulation (a) comparison between numerical (red line) and experimental values (blue line) of the force exerted on the structure; (b) wave evolution (t = 0.32 s) the wave colliding with the front of the obstacle; (t = 0.58 s) the wave wrapping around the structure, colliding together and continues moving toward the tank wall; (t = 1.44 s) the reflected wave hitting the back of the obstacle.

2. Another point of the validation part regards why the authors do not show the time history of the impact force. Since one of the characteristics of a debris flow impact process is its dynamic changes in time as the experiments of Song et al. 2021 show (the time history is quite complex and is not only represented by a single value!), the "simple" peak value is not sufficient for validating the model used. For this reason, I think that the authors' response "It is demonstrated that the RNG and GMO coupled model in FLOW-3D are able to describe the peak impact force and fluid surface effectively" is not fully trustable.

AUTHORS RESPONSE: Thanks for Reviewer's good suggestion! As responded in the comment #1, a new model validation has been executed, and the time history of impact force was determined to be compared, as shown in Fig. 2a.

3. A third comment regards the author's answer to comment #6 in combination with #12. I know well that long simulations use a high quantity of memory and take long computational times, so for this reason it could be, in some cases, acceptable to use high fixed discharge for a short time. However, I think that the 10-second duration used by the authors is not fully appropriate at least for some of the simulations used. For example, it is clear from Figure 17 that for the simulation with 45° of orientation (Or45) the peak impact force is the last value of the plot (i.e. at 10 second, so at the end of the simulation) but the force has a trend is still increasing! Also for the cases of 60° and 30° (i.e Or60 and Or30), the trend of the force is still increasing and at the end of the

simulation (i.e. the end of the plot), the values are very close to the peak values. This means that, at least, in these three simulations (but I think that the same problem arises also in lots of other simulations done by the authors as the one shown in Figure 21) the authors have to increase the time of the simulation until a significant (a few seconds?) decreasing, or at least constant, value of the impact force is visible.

AUTHORS RESPONSE: Thank you very much! We apologize for Reviewer's confusion induced by our unclear responses. In this study, the time of 10 s was not referred to the duration of debris flow hydrograph, but the computation time in FLOW-3D, that was 10 s after the debris flow was released from the inflow point. During this time, the debris flow head was ensured to move to the edge of deposition fan and the target building was fully exposed. It is important to emphasize that, therefore, the peak impact force involved in this study was the maximum value limited in the computation time of 10 s under a fixed discharge of 500 $m^3 s^{-1}$ (as added in Line 170-174 of the revised manuscript). As Reviewer mentioned, the peak impact force maybe not the maximum within a whole hydrograph. This is a very good suggestion, the longer simulation time will be taken into consideration in the future research.

4. The last comment regards the author's response to comment #9 regarding the force. For me, it remains unclear the meaning of the number that represents the force. Moreover, since the target building is a complex geometry, where these "numbers" are applied? It is quite different if this "number" is applied only to a single surface (e.g. only on the wall with the opening) or it is applied with different values on different surfaces (e.g. on the wall with the opening plus the roof) because the possible consequences are completely different! Here, I speak about "number" since, as in comment #9, I underline that the force is a vector, so a simple "number" does not represent the force: it is still missing the direction of this force and the point where the force is applied!.

AUTHORS RESPONSE: Thank you very much! We apologize for Reviewer's confusion induced by our unclear responses. Reviewer's comment on the overall impact force of target building can be concluded as two questions, specifically the meaning of the force number and position of force application. On the first question, the impact force is the combined hydraulic force due to the normal pressure and shear stress in the space system. With the help of GMO model in FLOW-3D, the normal pressure and shear force of the impacted object in *X*, *Y* and *Z* directions can be calculated at each time step (as added in Line 118-120 of the revised manuscript). On the second question, due to the complex geometry and variable built environments, the impacted elements of target building are changing in the different scenarios. For example, the wall B was impacted mainly in the orientation of 0° (Fig. 15a), and the wall A was impacted mainly in the Or90 scenario (Fig. 15e). In this study, therefore, the target building was treated as a whole bearing structure to keep consistency of analysis, that was the structural components, including the column, beam and bearing wall, were not analyzed separately. All over the grids covering building surface would be calculated when contacting with the flow (as added in Line 120-123 of the revised manuscript). As above-mentioned, the overall impact force is referred to the magnitude of combined fluid force, which is calculated from the all meshes covering the target building surface when impacted with the fluid. As Reviewer suggested, however, it is quite different damage state when the same magnitude of impact force is applied on the different position. This is a very good suggestion, we will take it into consideration in-depth in the future research.

[1] Gomez-Gesteira, M., Dalrymple, R. A.: Using a three-dimensional smoothed particle hydrodynamics method for wave impact on a tall structure. J Waterw Port Coast, 130, 63-69, https://doi.org/10.1061/(ASCE)0733-950X(2004)130:2(63), 2004.
[2] Liu, C., Yu, Z., and Zhao, S.: A coupled SPH-DEM-FEM model for fluid-particle-structure interaction and a case study of Wenjia gully debris flow impact estimation, Landslides, 18, 2403-2425, https://doi.org/10.1007/s10346-021-01640-6, 2021.