RESPONSE TO EDITOR AND REVIEWERS' COMMENTS:

Authors' General comment: Thank you for the Editor and Reviewers' 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 your professional opinions on the debris flow hazard. Reviewers' comments are all valuable and very helpful for revising manuscript and improve the future 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.

[RC1 and RC3 from Referee #1]

REVIEW COMMENTS:

[RC1] 1. Debris flow is a word commonly used to describe the flow of a mixture composed of water and a high concentration of sediment. Depending on the type and quantity of finer sediment grains, debris flow can be dived into two main classes: stony debris flow where the percentage of cohesive material (usually clay and the finer classes of silt) is negligible and mudflow where the percentage of cohesive sediment is important. This division is necessary since the rheological properties of the two are quite different: in the mudflow, the solid particles are essentially suspended inside fluid and the division between solid and fluid are difficult. This means that a monophase approach can be used and the mixture presents a non-Newtonian behaviour where yield stress is present (e.g., Bingham fluid). On the other hand, in the stony debris flow the two phases are easily identified and divided. This implies that a two-phase approach is necessary where the fluid behaviour is usually Newtonian, while the solid phase presents a collisional regime. See e.g., Iverson 1997, Takahashi 2007, Armanini 2013. <u>The authors have to clarify which type of debris flow are dealing with</u>: it seems, from the validation test that a stony debris flow is the target, however, in all the other sections it seems that a mudflow is analysed.

AUTHORS RESPONSE: We greatly appreciate for Reviewer's good comments! As Reviewer suggested, the division about which type of debris flow involved is indeed very necessary. We added the statements about debris flow type in Line 114-116 of the revised manuscript.

Line 114-116:

From the characteristics of RNG k-ε model, the type of debris flow involved in this study was determined as mudflow or viscous debris flow, in which a single-phase fluid was assumed and solid particles were treated as suspension and mixed with the fluid phase well.

[RC1] 2. The authors use the FLOW-3D code to simulate debris flow. <u>Which are the equations used</u>? <u>These are crucial when you describe the parameters used</u>. Without the equations, the parameters described by the author can be not present in the model. Moreover, looking at other papers dealing with FLOW-3D, also other parameters are needed: these are not listed in the manuscript.

AUTHORS RESPONSE: Thanks for Reviewer's good suggestion! In this study, the RNG k-ε model of FLOW-3D was applied to simulate the transportation process of debris flow. As Reviewer suggested, the statements about RNG model descriptions, equations and main parameters was added in Line 96-101 and Line 105-112 of the revised manuscript.

Line 96-101:

In this study, the renormalized group (RNG) model-based k-ε turbulence model and the general moving objects (GMO) model are applied to build fluid-solid coupled model of the debris flow impact. The RNG k-ε model is a modification of the standard k-ε model, which takes the turbulent vortex into account and provides an analytic formula for Prandtl number, as well as an analytic formula for low Reynolds number flow viscosity (Franco et al., 2021). These features make the RNG model more reliable and accurate for a broader flow than the standard k-ε model (Yin et al. 2015).

Line 105-112:

The two transport equations of RNG k-ε model in FLOW-3D are as following:

$$\frac{\partial k_T}{\partial t} + \frac{1}{V_F}\left\{uA_x\frac{\partial k_T}{\partial x} + vA_y\frac{\partial k_T}{\partial y} + wA_z\frac{\partial k_T}{\partial z}\right\} = P_T + G_T + Diff_{k_T} - \varepsilon_T \quad (1)$$

$$\frac{\partial \varepsilon_T}{\partial t} + \frac{1}{V_F}\left\{uA_x\frac{\partial \varepsilon_T}{\partial x} + vA_yR\frac{\partial \varepsilon_T}{\partial y} + wA_z\frac{\partial \varepsilon_T}{\partial z}\right\} = \frac{CDIS1\cdot\varepsilon_T}{k_T}(P_T + CDIS3\cdot G_T) + Diff_\varepsilon - CDIS2\frac{\varepsilon_T^2}{k_T} \quad (2)$$

where k_T is the turbulent kinetic energy, V_F is the fractional volume open to flow, A_x , A_y and A_z are the fractional area open to flow in the x, y and z directions, respectively. P_T is the turbulent kinetic energy production term, G_T is the buoyancy production term, *Diff* is the diffusion term, and ε_T is the turbulence dissipation term. In the RNG model of FLOW-3D, *CDIS*1 and *CDIS*3 are dimensionless user-adjustable parameters that have defaults of 1.42 and 0.2, respectively, and *CDIS*2 is determined from k_T and P_T (Flow Science, Inc., 2014).

[RC1] 3. A peculiarity of stony debris flow is the rapid formation of large scour and deposition. Deposition rapidly occurs when the mixture flow decreases the velocity, while scour usually happens when the flow is accelerated. Both decreasing and increasing velocities are present when the flow impacts the building. From the literature, I found that the model FLOW-3D can describe scouring and deposition for river and coastal morphodynamics, so when the morphological variation presents a longer time scale than the hydrodynamic one. In debris flow, the variation is of the same time scale. How did the authors consider this bed variation in the FLOW-3D modelling?

AUTHORS RESPONSE: We thank Reviewer very much for the comments! As Reviewer suggested, the formations of large scour and deposition indeed have the significant effects on the impact forces of debris flow. As mentioned in Question 1 (RC1), the type of debris flow involved in this study was mudflow or viscous debris flow, in which the particles were treated as suspension and mixed with the fluid phase well. The division between solid and fluid was assumed to be difficult, therefore, the granular deposition was not considered in this study (as added in Line 116-117 of the revised manuscript). This deposition fan was treated as a rigid bed model, therefore, the bed material scour would not happen in this study. To sum up, the bed variation induced by sediment transport processes, including sediment deposition and bed scour, was not considered (as added in Line 160-162 of the revised manuscript).

In addition, as Reviewer and literature mentioned, the scouring and deposition for river and coastal morphodynamics can be indeed calculated using the Sediment Scour model, another physics model implemented in FLOW-3D code. They are done by considering two states in which sediment can exist: suspended and packed sediment. Suspended sediment is typically of low concentration and advects with fluid flow. And only a thin surface layer of grains of the packed sediment (in the thickness of a few grain diameters) can move in the form of bed-load transport (Flow Science, Inc., 2014). In order to maintain the computation stability, moreover, the sediment diameter should not exceed 10% of cell size. If a smaller grid is usually applied to describe the building geometry precisely, the maximum value of sediment diameter will be tiny accordingly. Obviously, the above-mentioned requirements limit the availability in realistic debris flow stimulation, to a certain extent.

[RC1] 4. How are the impact forces evaluated? The authors write "the General Moving Objects (GMO) model of FLOW-3D was applied to obtain the overall impact forces on the building, in which a rigid body motion was introduced for the fluid-rigid interaction behaviour (Postacchin, 2019; Isobe, 2021)". Is it correct that the object where the forces are evaluated must be in motion? How is it possible to use this when dealing with a fixed and non-deformable target building as the one described in the manuscript? Moreover, the citations proposed are not relevant: Postacchini 2019 deals with experimental apparatus where a movable reference system is used (they move the building in a static pool of water), while Isobe 2021 deals with movable and deformable steel frame buildings but with another kind of models, not the FLOW-3D.

[RC1] 9. The force is a vector, so it has an orientation. In the paper, I suppose, the authors report only its module. This aspect gives rise to two main questions. The first one is how the impact force is evaluated: is it evaluated also considering the tangential stresses on the walls? The second question is about where the force is evaluated: it is all over the surfaces of the building (inside and outside walls)? If the answer is yes, is it simply a sum of the force exerted by the mixture over all the walls? In this specific case, if there is flow inside the building, is the force on one wall the net force evaluated between inside and outside or is it the sum of the two? Moreover, is it considered also the roof?

[RC3] 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. In addition, the former irrelevant citations have also been deleted.

Line 118-125:

With the help of GMO model in FLOW-3D, the combined hydraulic force due to normal pressure and shear stress can be calculated in the space system. The normal pressure and shear force of an impacted object in x, y and z directions can be gained at each time step. Due to the complex geometry and variable built environments, the impacted elements of target building were changing in the different scenarios. In this study, therefore, the target building was treated as a whole bearing structure to keep consistency of analysis. All over the grids covering building surface would be calculated when contacting with the flow. The GMO model can simulate the rigid body motion, which is either user prescribed or dynamically coupled with fluid flow. In this study, the target building and surrounding buildings were all prescribed to be the fixed and non-deformable rigid models.

[RC1] 5. In the validation section, <u>the authors reproduce one laboratory experiment</u>. <u>The particular stony debris flow experiments</u> can be reproduced well also with a monophase approach since the bed is rigid and all the material remains quite well mixed during all the experiment (only some separation between solid and fluid phase is visible in figure 2(c)). However, I think that it is not correct to say "FLOW-3D reproduces the debris flow impact process in the flume test very well" basing the statement mainly on the peak impact pressure. <u>It is important also the time history of the pressure</u>: arrival time of the flow, the timing of the peak, duration of the peak, etc. Moreover, <u>it is missing some parameters used in the model (e.g., the roughness)</u> and <u>it is not clear the dimension of the cell: is it composed of cubes of 0.02 m side</u>? If yes, since the first load cell position is 0.015 m from the bottom of the flume, <u>how do the authors evaluate the pressure at that height that is neither on the centre nor the border of the cube?</u> Additionally, on line 105 the <u>authors highlight that the data is averaged over 10 points (it means cells?) how is it possible to do this in the flume experiment? Is it horizontally averaged?</u> Finally, for better validation, I suggest using the calibrated parameter to reproduce a second flume experiment and discuss it.

[RC3] 1. The main one concern the validation. <u>In the laboratory experiment used for the validation, a well-mixed stony granular debris flow is reproduced</u>. 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.

[RC3] 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.

(1) Response to comment: "In the laboratory experiment used for the validation, a well-mixed stony granular debris flow (Song et al., 2021) is reproduced. And 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.

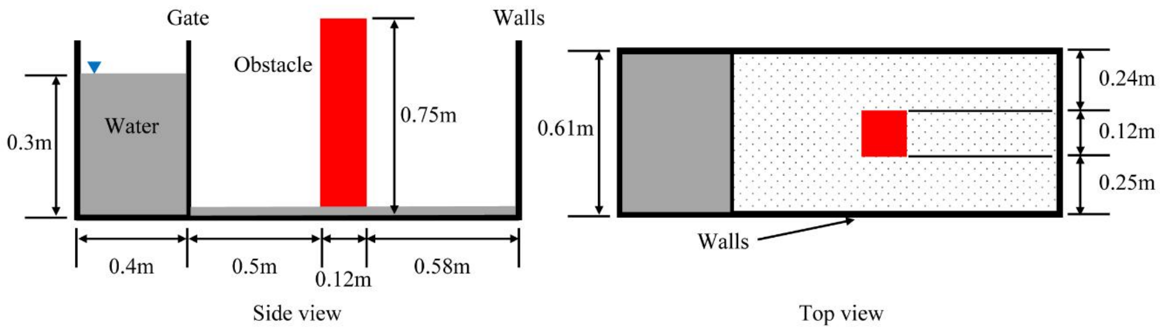

Figure 1. The geometric dimensions of the experimental dam-break model

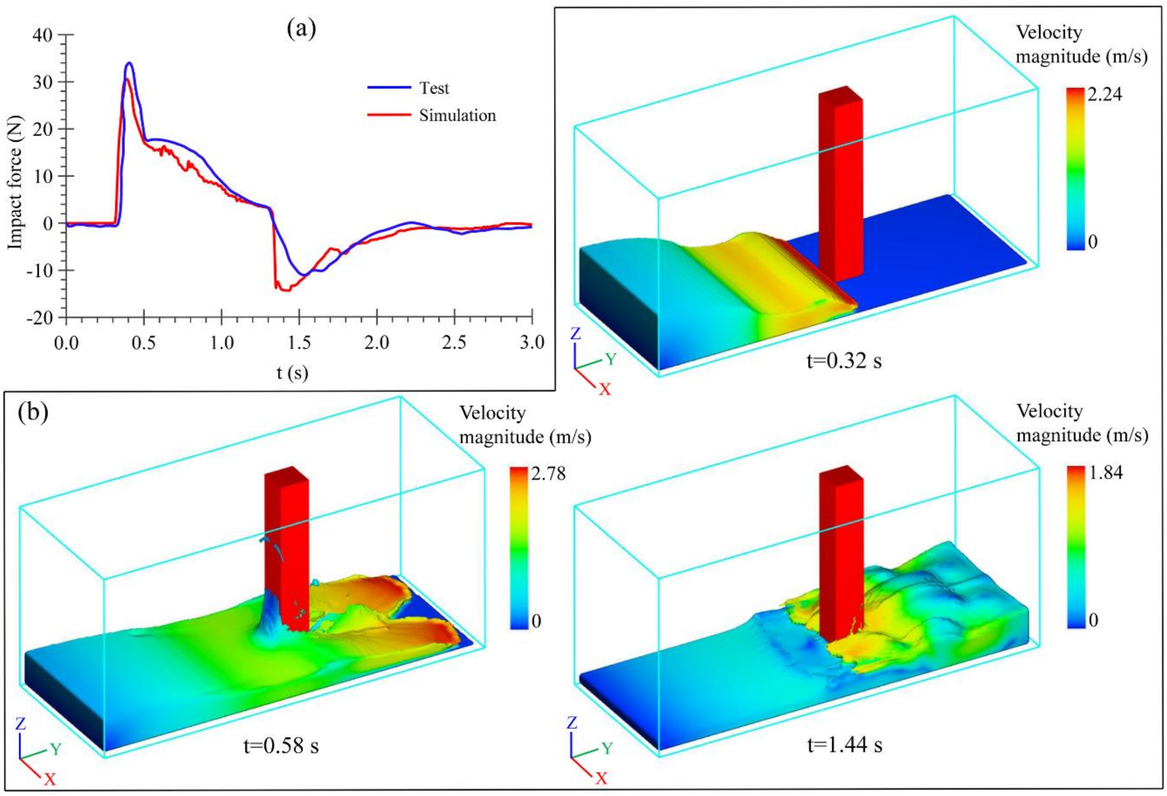

Figure 2. The dam-break simulation (a) comparison between numerical (red line) and experimental values (blue line) of the force exerted on the structure; (b) wave evolution (t = 0.32 s) the wave colliding with the front of the obstacle; (t = 0.58 s) the wave wrapping around the structure, colliding together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

(2) Response to comment: “It is important also the time history of the pressure. And why the authors do not show the time history of the impact force?”

AUTHORS RESPONSE: Thanks for Reviewer's good suggestion! As above-responded, 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) Response to comment: "it is missing some parameters used in the model (e.g., the roughness) and it is not clear the dimension of the cell: is it composed of cubes of 0.02 m side?"

AUTHORS RESPONSE: Thanks for Reviewer's comments! The statements about the surface roughness parameter of the flume bed were added in Line 162-163 of the revised manuscript. And the supplementary statement about the dimension of the computation cell was added in Line 138-139.

Line 162-163:

The surface roughness (k) of 0.05 m was set, meaning that the deposition fan surface was roughened with 5 cm diameter particles.

Line 138-139:

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.

(4) Response to comment: "how do the authors evaluate the pressure at that height that is neither on the centre nor the border of the cube?"

AUTHORS RESPONSE: We apologize for missing to describe the data collection approach of the load cells in FLOW-3D. The load cell was treated as a history probe in FLOW-3D. History probes are point measurement tools and can be thought of as thermocouples or pressure transducers, which can allow to access specific information at a particular location (Flow Science, Inc., 2014).

(5) Response to comment: "the authors highlight that the data is averaged over 10 points (it means cells?) how is it possible to do this in the flume experiment? Is it horizontally averaged?"

AUTHORS RESPONSE: We are very sorry for Reviewer's misunderstanding due to our inaccurate expressions. The impact force data was not averaged in the spatial dimension, but in the timeline. The raw impact force data was collected at interval of 0.001 s from the numerical code. To reduce the uncertainty, a simple data noise reduction approach, that the peak impact forces were obtained from the average values over 10 points in the timeline (0.01 s), was executed (Song et al., 2021). (as added in Line 125-127 of the revised manuscript).

[RC1] 6. In the numerical modelling, the authors used a fixed discharge of 500 m^3/s for a very short time (10 s). If the peak of discharge could be of some interest for very large debris flow, however, <u>the duration and the constant value are not realistic</u> and leads to unrealistic values of impact force. A more realistic debris flow inflow can be a triangular one where the overall duration is about 15 minutes with a peak discharge that occurs after 5 minutes (some examples of real and simplified hydrographs with can be found in Berger & al. 2011, Marchi & al. 2021). This modification in the inflow is essential for a truthful analysis of forces since, one of the main features of a debris flow just described previously, is the great deposition that occurs when the flow is slowed down. The direct consequence of the deposition is the time increase of the pressure due to this saturated terrain at rest.

[RC1] 12. I think that the duration of the simulation, which, from figures 17 and 21 it is set to 10 s (as the discharge duration), is too short since it for some tests the maximum value of the impact force is registered at the end of the simulation when a positive trend is also visible. I suggest increasing the simulation duration until the mixture is fully stopped or is flowed away from the target building.

[RC3] 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.

(1) Response to comment: “the duration and the constant discharge value are not realistic...”

AUTHORS RESPONSE: Thanks for Reviewer’s comments! As Reviewer mentioned, the realistic debris flow inflow discharge and duration are essential for a truthful analysis of impact forces. A 3-D numerical simulation with a realistic hydrograph, however, needs a large number of computer memory and processing time (as added in Line 165-166 of the revised manuscript). In this study, the computer memory of a single simulation is about 11 GB under the computation time of 10 s (as added in Line 188 of the revised manuscript), however, it will jump to 948 GB in the time of 900 s (a realistic debris flow duration of 15 min). In term of processing time, a flood FLOW-3D simulation with the duration of 1020 s (17 min) took a maximum of 425 hours (Gems et al., 2016). It is much hardly acceptable for the sensitivity analysis, in which the sufficient experimental groups are required (as added in Line 166-167 of the revised manuscript). Therefore, we set a time-invariance discharge of 500 m^3 s^{-1}, which can be of some interest for a large magnitude of debris flow (as added in Line 168-169 of the revised manuscript).

(2) Response to comment: “duration of the simulation is set to 10 s (as the discharge duration), is too short...”

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.

[RC1] 7. Some perplexity will arise also by looking at some of the parameters used: roughness and viscosity. For the surface roughness, the authors used 0.05 m which represents "the equivalent grain roughness (or absolute height in meters)". This means that on all the surfaces of the computational domain (that also includes the buildings) the roughness is generated by grains of 5 centimetres. This kind of roughness can be representative of a natural environment (e.g., riverbed, grassland, wood) but in an urban environment, where usually the surfaces are paved or made of gravel, is too big. For the viscosity, the authors used 1 Pas. This value is at least one order of magnitude higher compared to the ones described by Iverson 1997 (the fluid viscosity ranges from 0.001 Pas to 0.1 Pas) or also the ones measured by Song & al. 2021 (laboratory experiment with fluid viscosity ranging from 0.001 up to 0.1). Also, the authors use a value of 0.1 Pas to validate the FLOW-3D model (based on one of the experiments of Song & al. 2021). Why this choice? If you validate the model with 0.1 Pas, also the other simulations should be performed with similar viscosity.

AUTHORS RESPONSE: Thanks for Reviewer's comments! We apologize for the incomplete statements about the surface roughness and fluid viscosity. For the surface roughness, the roughness parameter was only set for the deposition fan surface. As Review suggested, the surface roughness (*k*) of 0.05 m was set, meaning that the deposition fan surface is roughened with 5 cm diameter particles, for the representation of the natural environment surrounding mountainous buildings (as added in Line 162-164 of the revised manuscript).

For the fluid viscosity, we apologize for missing to explain the reason for setting the different viscosity in the validation and analysis models. As Reviewer mentioned, the fluid viscosity ranges indeed from 0.001 Pa·s to 0.1 Pa·s in Iverson et al. (1997) and Song et al. (2021) 's physical experiments. However, it doesn't mean that the viscosity of realistic debris flow slurry could only be limited in this range. After the model validation, the availability of the RNG and GMO coupled model in capturing the peak impact force and fluid surface was demonstrated, to a certain extent. Theoretically, this model can also be applied in the cases of the other viscosity, except for the viscosity of 0.1 Pa·s. In order to compare with some realistic building damage cases, for example the Qipan gully and Zhouqu debris flows in the west of China, the rheological properties of numerical model was set as the viscous debris flow. Therefore, the debris flow density was set as 2000 kg m^{-3} and viscosity was empirically 1.0 Pa·s (Fig. 2.3 in Takahashi, 2007) (as added in Line 174-177 of the revised manuscript).

[RC1] 8. The target building has walls with a thickness of 0.35 m (line 156), while the cell (cube?) has a 0.25 m side (line 158). How is possible to simulate a wall that has a dimension that is not a multiple of a cell? Why not use a wall thickness equal to the cell side?

AUTHORS RESPONSE: Thanks for Reviewer's comments! We apologize for neglecting to clarify the reasons for setting cell size of the target building domain. The statements about the refining principles for the embedded mesh were added in Line 180-181 and Line 182-187 of the revised manuscript.

Line 180-181:

To maintain a balance between the computational accuracy and time cost, the whole computation domain was discretized at intervals of 0.5 m.

Line 182-187:

The embedded domain of target building should be refined further following two principles: (1) its cell size must be less than the wall thickness of 0.35 m. This is because the computation grids can not rotate following the changing orientations of target building. Once the building rotating, some lacks of building surface will be produced, due to the half of a single cell can not be covered by the wall element. (2) the boundaries of the embedded and external meshes must be overlapped for the computation stability, that is the external cell size is a multiple of embedded cell size. To sum up, the cell size of target building domain was determined as 0.25 m.

[RC1] 10. When the azimuth angle A decreases and approaches 0, it has to be specified that the two surrounding buildings become a single building. Regarding this aspect, is the metamodel able to consider this? Otherwise, the authors have to be neglected, from the metamodel simulations, all the cases when the surrounding buildings are merged.

AUTHORS RESPONSE: We greatly appreciate for Reviewer's comments! In this study, the metamodels were created from 160 simulation samples, in which the representative values of azimuth angle (***A***) were 0, 30, 45, 60 and 90, respectively. Only a single surrounding building was set when the azimuthal angle was 0° (as described in Line 206-207). To a certain extent, therefore, the merging effect of surrounding buildings was considered in the metamodel modeling.

[RC1] 11. Regarding the metamodel simulation, what are the ranges of variation of the four input variables?

AUTHORS RESPONSE: Thanks for Reviewer's good suggestion! The statements about the variation ranges of the four input variables were added in Line 225-227 of the revised manuscript.

Line 240-242:

In the metamodel modeling, the variation range of the orientation (***Or***) was from 0° to 90°, the opening scale (***Op***) was from 0 to 0.8, the azimuthal angle (***A***) was 0° to 90° and the distance (***D***) was from 5 m to 30 m.

Flow Science, Inc.: FLOW-3D v11.0.3 user manual, 2014.

Franco, A., Moernaut, J., Schneider-Muntau, B., Strasser, M., Gems, B.: Triggers and consequences of landslide-induced impulse waves-3D dynamic reconstruction of the Taan Fiord 2015 tsunami event, Engineering Geology, 294, 106384, https://doi.org/10.1016/j.enggeo.2021.106384, 2021.

Gems, B., Mazzorana, B., Hofer, T., Sturm, M., Gabl, R., and Aufleger, M.: 3-D hydrodynamic modelling of flood impacts on a building and indoor flooding processes, Nat. Hazards Earth Syst. Sci., 16, 1351-1368, https://doi.org/10.5194/nhess-16-1351-2016, 2016.

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.

Iverson, R. M.: The physics of debris flows, Reviews of Geophysics. 35, 245–296, https://doi.org/10.1029/97RG00426, 1997.

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.
Song, D., Chen, X., Zhou, G. G. D., Lu, X., Cheng, G., Chen, Q.: Impact dynamics of debris flow against rigid obstacle in laboratory experiments, Engineering Geology, 291, 106211, https://doi.org/10.1016/j.enggeo.2021.106211, 2021.
Takahashi, T.: Debris Flow Mechanics, Prediction and Countermeasures, Taylor & Francis Group, London, UK, 2007.
Yin, Y. P., Huang, B., Chen, X., Liu, G., and Wang, S.: Numerical analysis on wave generated by the Qianjiangping landslide in Three Gorges Reservoir, China. Landslides, 12, 355-364, https://doi.org/10.1007/s10346-015-0564-7, 2015.

[RC2 from Referee #2]

REVIEW COMMENTS:

According Martinez-Carvajal et al (2018), a natural phenomenon (hazard) may be characterized in terms of temporal, spatial and magnitude probabilities. The effects of the interaction between the hazard and the exposed element depend on the intensity of the hazard and on the resistance, sometimes called susceptibility, of the element at risk, which describes the propensity of a building or other infrastructure to suffer damage from a specific hazard impact. Consequently, a modern concept of vulnerability must consider the intensity of the hazard as well as the structural resistance of the exposed infrastructure. This concept is referred to as physical vulnerability, and the most accepted definition is a representation of the expected degree of loss quantified on a scale of 0 (no damage) to 1 (total destruction).

Previous considerations leads me to suggest to the authors the inclusion of a broad discussion on vulnerability which certainly is the major objective of this kind of research. Comments on the effect of the buildings strength will be profitable for opening future research topics by means of numerical modelling.

AUTHORS RESPONSE: We greatly appreciate for Reviewer's good comments! As Reviewer suggested, a broader discussion on the contributions of the present paper towards debris flow hazard and vulnerability assessment was added in Line 529-551 of the revised manuscript.

Line 529-551:

It is obvious that the quantitative descriptions about the interactions between the built environment and impact forces can be useful to the built environment improvement and local adaptation measures for the impact force reduction, which are assumed as the low-cost and efficient approach for mitigating the building's structural damages. And more significantly, the present paper has extended the knowledge about the influence factors on debris flow intensity. It is demonstrated that some artificial building factors can not be ignored, except for the natural environments, in deciding the spatial pattern of the process intensity. The further research about their relative importance with the 3-D numerical simulation and sensitivity analysis can promote the relative intensity evaluation of the building, especially in terms of the indicator selection and weighting, which may open a future topic of the debris flow hazard assessment. For the building vulnerability assessment, the indicators can be mainly divided into two kinds: the exterior process intensity and interior building resistance. The process intensity, for example the flow depth, velocity, impact force or the other proxy, was assumed absolutely necessary, either in the curve based approach or the indicator based approach (Martinez-Carvajal et al., 2018). From the current literature, however, there are some confusions in selecting the surroundings factors and process intensity indicator. To be specific, some surroundings factors or also called protection factors, including the *Surrounding buildings*, *Building row*, *Wall around building*, *Natural barriers* and so on, were still selected when the debris flow intensity had been indirectly considered (Dall'Osso et al., 2009; Dall'Osso et al., 2016; Papathoma-Köhle et al., 2019). These indicators should be independent each other theoretically. From the views of the present paper, the functions of all over the surroundings factors are the influences on the process intensity around building. Therefore, the process intensity should be exclusive with the surroundings factors. The building features factors are mainly considered to be acted on the building resistance, including the *Material*, *Structure*, *Number of stories*, *Foundation strength* and so on. However, It is not hard to find that some building indicators, for example the *Orientation*, *Shape* and *Openings*, can rebuild the process intensity. As a result, the effect of the representative building features indicators on the building vulnerability needs an in-depth discussion in future. The last but not least, a more universal, robust index may be developed using the numerical simulation approach, which can improve the locality limits resulting from the empirical data, to some extent.

Dall'Osso, F., Dominey-Howes, D., Tarbotton, C., Summerhayes, S., Withycombe, G.: Revision and improvement of the PTVA-3 model for assessing tsunami building vulnerability using ''international expert judgment'': introducing the PTVA-4 model, Nat Hazards, 83, 1229-1256, https://doi.org/10.1007/s11069-016-2387-9, 2016.

Dall'Osso, F., Gonella, M., Gabbianelli, G., Withycombe, G., Dominey-Howes, D.: A revised (PTVA) model for assessing the vulnerability of buildings to tsunami damage, Nat. Hazards Earth Syst. Sci., 9, 1557-1565, https://doi.org/10.5194/NHESS-9-1557-2009, 2009.

Martinez-Carvajal, H. E., de Moraes Guimaraes Silva, M. T., Garcia-Aristizabal, E. F., Aristizabal-Giraldo, E. V., Larios-Benavides, M. A.: A mathematical approach for assessing landslide vulnerability. Earth Sciences Research Journal, 22, 251-273. https://doi.org/10.15446/esrj.v22n4.68553, 2018.

Papathoma-Köhle, M., Schlögl, M., and Fuchs, S.: Vulnerability indicators for natural hazards: an innovative selection and weighting approach, Scientific Reports, 9, 15026, https://doi.org/10.1038/s41598-019-50257-2, 2019.