RESPONSE TO REVIEWER COMMENTS:

Authors' General comment: Thank you 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 believe the revised version of the manuscript, which addresses the Reviewer's comments, is now more consistent with the current literature and clarifies the important points raised by the Reviewer. Revised portions are marked in red in the manuscript and response letter, and the manuscript is re-submitted in clean format to the Journal. Please also find below my response to Reviewer comments.

REVIEW COMMENTS:

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. The authors have to clarify which type of debris flow are dealing with: 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 non-Newtonian fluid was assumed and solid particles were treated as suspension and mixed with the fluid phase well.

2. The authors use the FLOW-3D code to simulate debris flow. Which are the equations used? These are crucial when you describe the parameters used. 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:

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

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

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).

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, 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 176-178 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.

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.

AUTHORS RESPONSE: Thank you very much! We apologize for Reviewer’s confusion induced by our incorrect statements and references. The statements about GMO model description, building setting and impact force calculation approach were added in Line 119-122 and Line 123-125 of the revised manuscript. In addition, the former irrelevant citations have also been deleted.

Line 119-122:

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. The hydraulic force and torque due to normal pressure and shear stress can be calculated at each time step.

Line 123-125:

The overall impact force of target building was gained from the combined fluid force considering the force direction in the GMO model, which was calculated from the normal pressures and shear forces in x, y and z directions.

5. In the validation section, the authors reproduce one laboratory experiment. The particular stony debris flow experiments 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. It is important also the time history of the pressure: arrival time of the flow, the timing of the peak, duration of the peak, etc. Moreover, 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? If yes, since the first load cell position is 0.015 m from the bottom of the flume, how do the authors evaluate the pressure at that height that is neither on the centre nor the border of the cube? Additionally, on line 105 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? Finally, for better validation, I suggest using the calibrated parameter to reproduce a second flume experiment and discuss it.

(1) Response to comment: "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."

AUTHORS RESPONSE: We thank Reviewer for the very good advice! It is really true as Reviewer suggested that there are many indicators to describe the fluid impact process, except for the peak impact force. Therefore, the former statement that "FLOW-3D reproduces the debris flow impact process in the flume test very well" is indeed very biased. The revised statement is that "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." (as added in Line 171-172 of the revised manuscript)

(2) 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 156-157 of the revised manuscript. And the supplementary statement about the dimension of the computation cell was added in Line 153-154.

Line 156-157:

In the physical experiment, the flume bed was roughened using 0.6 mm spherical glass beads, its surface roughness parameter was set as 0.0006 m accordingly in the validation model.

Line 153-154:

the targeted analysis domain was discretized into a grid with a cell size of 0.02 m, which was equal to a cube of 0.02 m side in 3-D model.

(3) 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. As Reviewer mentioned, the load cell 1# was indeed set to be 0.015 m from the bottom of the flume, nearly the border of computation grid, however, this didn't affect its measured impact force. This is because that 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) (as added in Line 167-169 of the revised manuscript).

(4) Response to comment: "on line 105 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 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).

(5) Response to comment: "for better validation, I suggest using the calibrated parameter to reproduce a second flume experiment and discuss it."

AUTHORS RESPONSE: Thanks for Reviewer's good suggestion! As Reviewer suggested, multiple groups of validation simulations can indeed improve reliability of the numerical model, however, these are usually limited by the available data. In the Song et al. (2021) 's paper, only three groups of debris flow impact tests were published in detail, the test ID is 40-100-15, 50-100-15 and 55-100-15, respectively. There were two main reasons why the test ID of 50-100-15 was selected for model verification: (1) all the material remained quite well mixed with fluid phase, and a tiny separation between solid and fluid phase was visible during all the experiment, therefore, it was considered to be suitable for the RNG model validation. (2) the representative flip-through impact phenomenon was presented in this test (as added in Line 131-134 of the revised manuscript).

In the test group of 40-100-15, however, there was an obvious separation between solid and fluid phase when the flow was jetted vertically along the barrier face (see Fig. 5(a) and supplemental videos in Song et al. (2021)'s paper). In the test group of 55-100-15, static load by gradual filling of the subsequent debris was only demonstrated, due to the subcritical Froude condition, and the dynamic impact is not obvious (Song et al., 2021). It is obvious that the later two groups of experiments are not appropriate for the RNG model, and it is also proved by the poorer comparison results in terms of peak impact force, which were executed in our preliminary experiments.

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, the duration and the constant value are not realistic 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.

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 183-184 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 205 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 184-185 of the revised manuscript).

Therefore, we set a fixed discharge of 500 m^3/s, which can be of some interest for a large magnitude of debris flow (as added in Line 186-187 of the revised manuscript). It is important to correct that the time of 10 s is not the inflow duration, but the computation time. The debris flow was ensured to flow out of the deposition fan and the target building was fully exposed during this time. 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 with a fixed discharge of 500 $m^3 s^{-1}$ (as added in Line 189-191 of the revised manuscript).

For the pressure induced by the deposition as Reviewer mentioned, we are very sorry that it was ignored due to the limits of the RNG model. It is demonstrated that the deposition of debris in front of the building increases the impact pressure indeed, resulting from the active earth pressure of the water-saturated sediment (Gao et al., 2017). However, it seems to has little effect on the peak impact force. From the laboratory results, this part of forces can be regarded as an almost constant static pressure at the end of the experiments, which is substantially lower than the peak impact force (Sturm et al., 2018a).

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 178-180 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 192-194 of the revised manuscript).

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 197-198 and Line 199-204 of the revised manuscript.

Line 197-198:

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

Line 199-204:

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.

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?

(1) Response to Question 1: "how the impact force is evaluated: is it evaluated also considering the tangential stresses on the walls?"

AUTHORS RESPONSE: We apologize for missing to describe the computation approach of impact force. The overall impact force of target building was gained from the combined fluid force considering the force direction in the GMO model, which was calculated from the normal pressures and shear forces in x, y and z directions (as added in Line 123-125 of the revised manuscript). Therefore, the tangential stresses on the walls have been considered in this study.

(2) Response to Question 2: "where the force is evaluated: it is all over the surfaces of the building (inside and outside walls)?..."

AUTHORS RESPONSE: Thanks for Reviewer's comments! All over the surfaces of target building are stressed elements, including the exterior wall, interior wall and roof (as added in Line 122-123 of the revised manuscript). The overall impact force is not simply the sum of absolute force values from each stressed element, but the combined force considering the force direction. In our preliminary experiment, the two sides of a wall was designed to be impacted simultaneously under the same hydrodynamic conditions. The gained impact force was not twice as the only one-side impact, on the contrary, it was too small to be ignored. As Reviewer assumed, in a specific case where the inside and outside wall are impacted simultaneously, the overall impact force is definitely the net force value between the impact forces of the two walls.

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 221-222). To a certain extent, therefore, the merging effect of surrounding buildings was considered in the metamodel modeling.

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 240-242 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.

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.

AUTHORS RESPONSE: Thanks for Reviewer's good suggestion! As responded in Question 6, 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 with a fixed discharge of 500 m^3 s^{-1}. During this time, the debris flow was ensured to flow out of the deposition fan and the target building was fully exposed.

[1] 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.

[2] 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.

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

[4] 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.

[5] 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.

[6] Gao, L., Zhang, L. M., and Chen, H. X.: Two-dimensional simulation of debris flow impact pressures on buildings, Engineering Geology, 226, 236-244, https://doi.org/10.1016/j.enggeo.2017.06.012, 2017.

[7] Sturm, M., Gems, B., Keller, F., Mazzorana, B., Fuchs, S., Papathoma-Köhle, M., and Aufleger, M.:

Understanding impact dynamics on buildings caused by fluviatile sediment transport, Geomorphology, 321, 45-59, https://doi.org/10.1016/j.geomorph.2018.08.016, 2018a.
[8] Iverson, R. M.: The physics of debris flows, Reviews of Geophysics. 35, 245–296, https://doi.org/10.1029/97RG00426, 1997.
[9] Takahashi, T.: Debris Flow Mechanics, Prediction and Countermeasures, Taylor & Francis Group, London, UK, 2007.