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, please find below my response to Reviewer's comments.

REVIEW COMMENTS:

1. I know very well that the longer the simulation, the longer the time used and the bigger the amount of data to be analysed, but I think it is not entirely acceptable that all the analysis is developed with a limited time for the evaluation of the maximum peak impact force. As mentioned in [RC1] and [RC3] for all the two cases in which the time variation of the impact force is reported (so Figure 17 and Figure 21) the force trend is still increasing at the end of the plot (simulation) and some maximum values are evaluated exactly at the end. Additionally, some intersections between the impact force are present depending on the variable analysed. This means that if the simulation is longer than the used 10s and since the maximum values can increase, it can happen that also the analysis of the variable involved can change giving rise to a modification of the effect: by way of example only, it can happen that if the simulation is longer, the peak impact force of the case Or30-Op0-Anull-Dnull becomes greater than Or0-Op0-Anull-Dnull so all the discussion provided by the author present a limited scientific meaning. This means that lines 172-174 were not sufficient as an explanation for the time length, nor the statement that a longer simulation requires too much time.

I suggest that in at least one of the tests where the time variability of the impact force is plotted, the authors produce longer simulation (for example, as suggested in [RC3] "increase the time of the simulation until a significant (a few seconds?) decreasing, or at least constant, value of the impact force is visible") in order to show that the used value produce something that has real scientific relevance. I think that providing this for at least one involved variable will demonstrate that the peak impact force values used are trustworthy.

AUTHORS RESPONSE: We greatly appreciate for Reviewer's good comments! In the previous simulations, the debris flow discharge at the inflow cross section was set to be fixed (500 $m^3\ s^{-1}$), and the duration of hydrograph was not limited. The debris flow, in other words, was released endlessly from the inflow cross section. The previous computation time of FLOW-3D model was set 10 s, and the peak impact force was considered as the maximum value in this time range. As Reviewer suggested, however, it's hardly to keep the impact process stable. It is found that some peak impact force, therefore, was evaluated at the end of simulation, when the force trend was still increasing. In order to address the problem completely, **the duration of debris flow hydrograph and computation time of FLOW-3D model for all the simulation scenarios are reconsidered in the revised manuscript.** Specifically, the debris flow discharge is still fixed as 500 $m^3\ s^{-1}$, and the duration time is set 5 s at the inflow cross section. In consideration of the distance between inflow cross section and target building, the entire simulation computation time is extended to 15 s to ensure that nearly all the debris flow can flow through the target building (as described in Line 176-182 of the revised manuscript). During this time, there are the distinct rise and fall trends in the impact process line, to ensure the peak impact force is evaluated more scientifically. And all the relevant simulation data and analysis results have been updated in the revised manuscript (as marked in red).

2. The title can be misleading. The paper tackle monophasic viscous debris flow with a (very) synthetic and simplified hydrograph. I think that the underline words, or their meaning, must appear in the title. These concepts must appear also in the Abstract.

AUTHORS RESPONSE: Thanks for Reviewer's good suggestion! As Reviewer suggested, the title has been updated as "*Sensitivity analysis of a built environment exposed to a synthetic monophasic viscous debris flow impacts with 3-D numerical simulations*". And the Abstract has also been revised accordingly, as added in Line 15-16 of the revised manuscript.

Line 15-16:

The impact forces were obtained from the monophasic viscous debris flow with a synthetic and simplified hydrograph using the FLOW-3D model...

3. Line 12: "peak impact forces" -> It is not completely true since is the value obtained by a limited-time simulation of 10s.

AUTHORS RESPONSE: We greatly appreciate for Reviewer's good comments! As described in Question 1#, the modified computation time has been extended to be triple of the inflow duration, in order to ensure nearly all the debris flow can flow through the target building. The peak impact force, therefore, is treated as the true maximum value in a relatively complete impact process (as added in Line 182-183 of the revised manuscript).

4. Line 105: "transport equations" -> It is formally correct to say that equations (1) and (2) are the transport equation of the RNG k-ε model. However, some readers used to deal with sediment moving over the fixed bed and/or flow tracers (e.g., chemical components advected by a flow field) can be confused. I think it is better to say that the equations describe the turbulent kinetic energy (k_T) and the turbulence dissipation (ε_T) balance equations of the RNG k-ε model.

AUTHORS RESPONSE: Thanks for Reviewer's good suggestion! As Reviewer suggested, the statement of "transport equations" has been revised as "turbulent kinetic energy and the turbulence dissipation balance equations" (as added in Line 108 of the revised manuscript).

5. Line 147 "perfectly" -> If the numerical model perfectly reproduces something, no difference with experimental results must be present and this is not the case. Please modify.

AUTHORS RESPONSE: Thanks for Reviewer's good suggestion! As Reviewer suggested, the statement of "perfectly reproduced" is indeed too absolute. This statement is modified in Line 152-154 of the revised manuscript.

Line 146-148:

Fig. 2a shows the general 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 reasonably reproduced by the numerical model.

6. Lines 162-164 -> The authors describe the values used for the roughness of the inclined plane where the viscus debris flow moves, but what about the building? Are they without roughness, so are they treated as smooth surfaces? Please justify the choice. Moreover, it is missing the roughness used in the model validation.

AUTHORS RESPONSE: We greatly appreciate for Reviewer's good comments! Firstly, the surface roughness of the obstacle is not determined in the physical test. Therefore, k_o (0, 0.001, 0.002, or 0.003 m) is selected for sensitivity analysis to determine if this parameter could reasonably reflect the macro mechanical behaviors of the dam-break test (as described in Line 141-143 of the revised manuscript). It is indicated that the impact force is very low sensitivity to k_o in Fig.2a, this study does not pay too much attention to the value of this parameter, and the surface roughness (*k*) of all the impacted object in numerical simulation is set 0 m (as described in Line 154-156 of the revised manuscript). Secondly, the tank is considered to be smooth surface in the model validation, and its surface roughness (k) is set 0 m (as added in Line 137-138 of the revised manuscript).

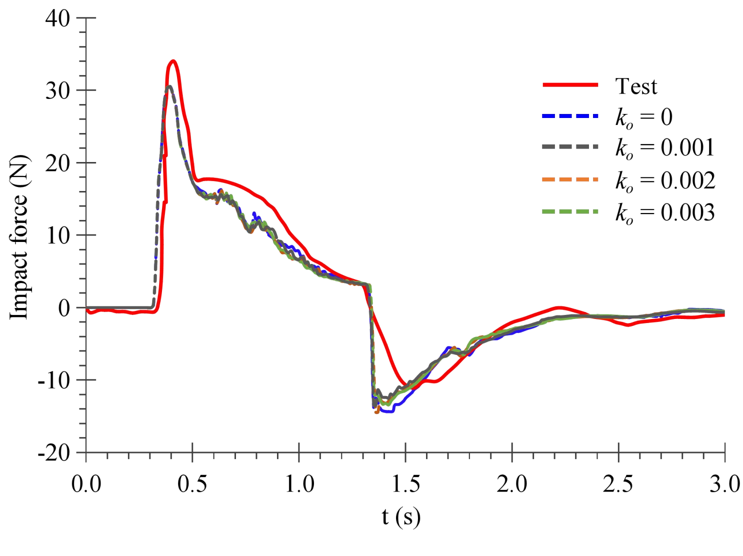

Figure 2a. The dam-break simulation comparison between numerical (dotted lines) and experimental values (red line) of the force exerted on the structure.

7. Lines 167-174 -> From the text it seems that the discharge, that feeds the computational domain, is considered constant and equal to 500 m^3/s for all the duration of the simulations (the authors wrote on lines 168-169 "an inflow with a time-invariance discharge of 500 m^{3}s^{-1}" and on lines

173-174 "in the computation time of 10 s under a fixed discharge of 500 m^3s^{-1}"). However, in the response to the reviewer, the authors highlighted that "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" that seems something completely different from the previous statement. From this answer, I understand that the simulation time is 10 s, but the discharge of 500 m^3s^{-1} has a duration completely different and it seems also that this discharge occurs only for a few time steps (or only in the first one). Please clarify this aspect because it is fundamental for understanding the work: the general meaning of the work is completely different if the discharge lasts for 10s or for only some time steps.

AUTHORS RESPONSE: We greatly appreciate for Reviewer's good comments! We apologize for Reviewer's confusion induced by our unclear statements. In the newly revised manuscript, as described in Question 1#, the duration of 500 m^3 s^{-1} discharge is adjusted as 5 s at the inflow cross section. In consideration of the distance between inflow cross section and target building, the entire computation time is extended to 15 s, the triple of inflow duration, to ensure that nearly all the debris flow can flow through the target building. During this time, there are the distinct rise and fall trend in the impact force time history line, and the peak impact force is treated as the true maximum value in a relatively complete impact process (as added in Line 176-183 of the revised manuscript).

8. Lines 182-187 -> For me it is not fully clear how the wall dimension can be 0.35m. If I understand well the text, the domain associated with the building is discretized with cells of 0.25m (that is half the size of the cell used in the rest of the computational domain). But if the dimension of the building cells is 0.25m, how is it possible to have a wall dimension of 0.35m? Is the 0.35m value an average value between the different configurations (I understand that the cell could not rotate, so in certain cases, the wall is discretized by, for example, 2 cells)? Or is there present some numerical procedure that can subdivide the building cells into fractions? Please clarify this point.

AUTHORS RESPONSE: We thank Reviewer very much for the comments! Yes! Just as Reviewer assumed, FLOW-3D code can subdivide the building cells into fractions. The FLOW-3D procedure uses an unique FAVOR^{TM} technique, an acronym for *Fractional Area/Volume Obstacle Representation*, to computes the open area fractions (AFT, AFR, AFB) on the cell faces along with the open volume fraction (VF) for reconstructing the geometry for a simulation. This approach offers a simple and accurate way to represent complex surfaces in the domain without requiring a body-fitted grid. The FAVOR processor can generate area fractions for each cell face in the grid by determining which corners of the face are inside of a defined geometry, and incorporate geometry effects into the governing equations (as added in Line 96-97 of the revised manuscript). If all four corners of a cell face are inside the geometry, then the entire face is defined to be within the geometry (as illustrated the red cell in Fig.8-1). Similarly, if all corners lie outside, then the entire face is assumed to be outside the geometry (as illustrated the circle in the lower right corner of the mesh). When some face corners are inside a geometry and some are outside, the intersection of the geometry with face edges are computed. Area fractions are then computed from these intersection points assuming straight-line connections between intersection points within the face (as illustrated the blue cell).

However, for some geometries and mesh resolutions it is possible that the geometry may intersect a cell face more than once. In this case the corresponding cell edge is assumed to be either fully inside the object or fully outside (as illustrated the orange cell). This may happen in case of building rotating of the present study, as illustrated in Fig.8-2, the wall geometry can not be captured accurately (as added in Line 191-193 of the revised manuscript). The representation is improved as the mesh resolution is increased (i.e., the cell size is decreased). When the cell size is decreased from 0.35 m to 0.25 m, as shown in Fig.8-3, the geometry recognition accuracy has been greatly improved.

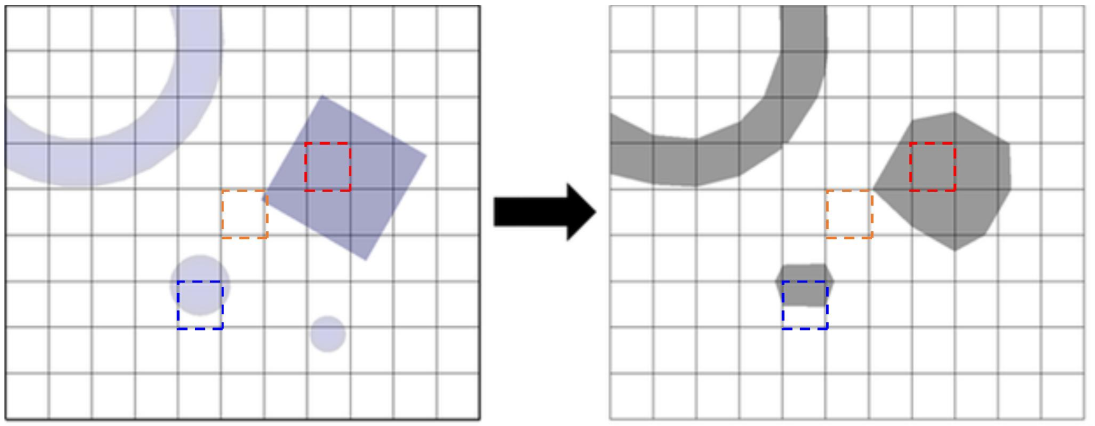

Figure 8-1. Object definition (left) and object created (right)

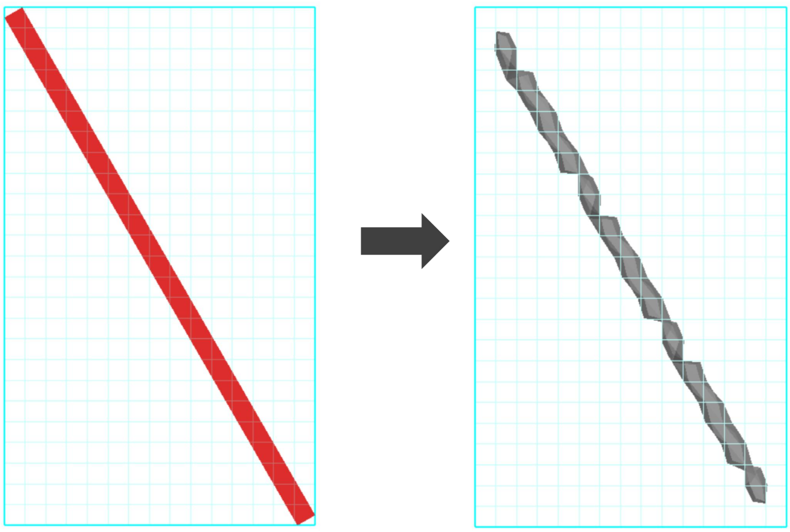

Figure 8-2. Object definition (left) and object created (right)

(The red wall is rotated counterclockwise by 30°, the wall thickness is 0.35 m, the blue cell size is 0.35 m)

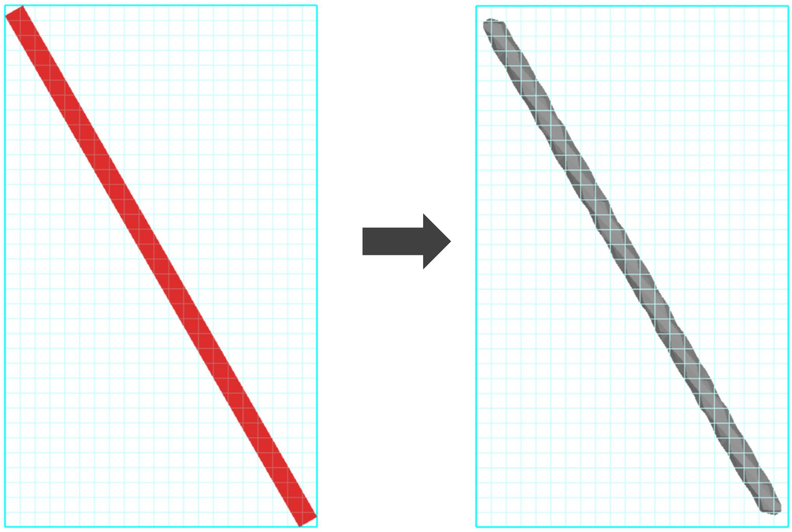

Figure 8-3. Object definition (left) and object created (right)
(The red wall is rotated counterclockwise by 30°, the wall thickness is 0.35 m, the blue cell size is 0.25 m)