Applied Numerical Modeling in Geomechanics – 2020 – Billaux, Hazzard, Nelson & Schöpfer (eds.) Paper: 14-02
©2020 Itasca International, Inc., Minneapolis, ISBN 978-0-9767577-5-7

Simulation of impacts on a rockfall protection wall made of interconnected concrete blocks

A. Furet[1,2,3], S. Lambert[2], P. Villard[3] & J.P. Jarrin[1]
[1] *Geolithe, 181 rue des Bécasses, 38920 Crolles, France*
[2] *Univ. Grenoble Alpes, Irstea, ETNA, 38000 Grenoble, France*
[3] *Univ. Grenoble Alpes, CNRS, Grenoble INP, 3SR, 38000 Grenoble, France*

1 INTRODUCTION

Rockfall protection embankments are efficient to protect elements at risk against rockfall. However, these structures present the inconvenient of a large footprint. In addition, considering the current economic and societal needs, protection structures have to be constructed in a limited time with a minimum impact on the traffic. In this context, the innovative structure Bloc Armé®, made of concrete blocks and steel reinforcement elements, was developed to allow to quickly rise a protection with low footprint.

During the past 30 years, many studies have been conducted for investigating the response of impacted embankments and their capability to withstand the impact (Lambert et al. 2014) and some numerical studies often complete the experiments (Peila et al. 2007, Breugnot et al. 2015). However, in France and other countries, no official regulations exist for the design of protective structures subjected to high energy rock impact. The different analytical methods proposed for earth embankments, suggesting for example to estimate the block penetration, are not valid for the specific Bloc Armé® structure. The dynamic nature of the loading, simplified in the analytical methods, should be considered in the structures design. Moreover, taking into account the mechanisms involved during impact represents a possible improvement for the development of design methods. Considering the complexity of the structure and the dynamic loading, the development of a 3d numerical model is necessary. The objectives of the numerical development are twofold: provide a better understanding of the physical phenomena, in particular by addressing the energy dissipation, and lead to design tools for engineering.

For this purpose, a detailed 3d model of the structure is developed with *FLAC3D* (Itasca 2017). FLAC3D is chosen because this software allows to develop a model for engineering application while providing advanced features to produce a specific complex model, execute dynamic calculations and accept large displacements. Many experimental data are available for the determination of the physical parameters and the model calibration. Part of the values used for the physical parameters, the model calibration and the simulations validation are based on data from specific experiments. Two experimental test campaigns involving impact on such structure were conducted. A first one at the real scale, for which blocks with a length of 1.6 m and a width and a height of 0.8 m were used, and a second one at small scale, in a size ratio of ¼. These latter tests allowed testing different wall designs, including the presence of perpendicular shear walls. In addition, parameters concerning the concrete and the interfaces are determined thanks to static tests.

After the presentation of a simplified model of a small-scale linear wall, we present here the preliminary impact simulation results compared to the experimental tests results before exposing the perspectives concerning the numerical modeling of the structure.

2 STRUCTURE AND MODEL DESCRIPTION

The structure is composed of concrete blocks linked to each other by means of steel continuous reinforcements in the form of bars and plates (Fig. 1). In the vertical direction, the bars are introduced through holes in the blocks and through transversal plates placed on the block's surfaces. The horizontal plates connect contiguous blocks. Even though the blocks are connected by rigid elements, the gaps between elements give the structure a certain deformability.

In the numerical model, the concrete blocks are modelled by discrete blocks of continuum media stacked to form the wall. The metallic reinforcements are modelled thanks to structural elements existing in *FLAC3D*. The contacts between concrete blocks are managed thanks to so-called interfaces.

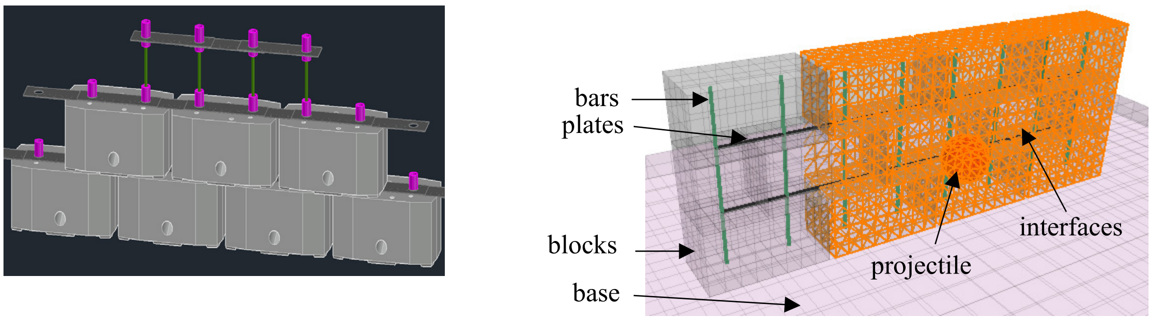

Figure 1. Bloc Armé® a) drawing of the real structure b) numerical model.

2.1 *Constitutive models*

The main objective of the model is to reproduce the global structure behavior in terms of displacement. Consequently, the constitutive model for the concrete blocks is kept simple. Nevertheless, the energy dissipation associated to damage is accounted for through a plastic model. The existing Mohr-Coulomb constitutive model is chosen in this purpose. The failure envelope of this model, combining a Mohr-coulomb criterion and a limit in tension, allows accounting for the concrete damage.

Some of the Mohr-Coulomb law parameters are determined from static tests on cylindric samples while others are determined from the literature. Young modulus E_{block} is given by achievement of uniaxial compressive tests, whereas the tension limit σt_{block} is given by the achievement of Brazilian tensile tests. The internal friction Φ_{block} and the cohesion C_{block} are determined from the cumulated results of the two tests.

Elastic constitutive law is chosen for the base and the projectile which are not exposed to plastic strain. The mechanical properties of the three materials are given in Table 1.

Table 1. Mechanical properties of the continuous media for the small-scale experiment materials.

	Blocks		Projectile		Base	
Elastic modulus (Pa)	E_{block}	5.0×10^9	E_{proj}	1.0×10^{10}	E_{base}	5.0×10^9
Poisson's ratio (−)	ν_{block}	0.2	ν_{proj}	0.2	ν_{base}	0.2
Internal friction angle (°)	φ_{block}	55	φ_{proj}	-	φ_{base}	-
Cohesion (Pa)	C_{block}	5.9×10^6	C_{proj}	-	C_{base}	-
Dilatancy angle (°)	ψ_{block}	0	ψ_{proj}	-	ψ_{base}	-
Tension limit (Pa)	σt_{block}	3.7×10^6	σt_{proj}	-	σt_{base}	-

2.2 *Interfaces*

Interfaces are used for managing contacts of the concrete blocks and the projectile, allowing sliding, detachment and force transfers. Cohesion, friction and strength limits are chosen to physically represent the behavior of each interface. The friction angle at the block/block and block/base interfaces have been determined by static sliding tests.

For the normal and tangential stiffness, high values are considered in order to limit the influence of elastic strain of the interfaces. The values of the normal stiffness are defined using the formula proposed in Equation 1 (Itasca 2006) and the value is divided by 10 to obtain the tangential stiffness.

$$kn = \max\left(\frac{K + \frac{4}{3}.G}{\Delta zmin}\right) \tag{1}$$

where *K* and *G* are the bulk and shear modulus of materials in contact and *Δzmin* is the smallest dimension of an adjoining zone in the normal direction

Interfaces are placed on each face of the blocks, at the base surface and on the projectile surface. Having two interfaces in contact allows limiting contact detection problems and reducing uncertainty associated with the blocks boundaries contacts. The mechanical properties of the interfaces are presented in Table 2.

Table 2. Mechanical properties of interfaces for the small-scale experiments.

	block/block	block/base	block/projectile
Friction angle (°)	23	28	30
Normal stiffness, kn (Pa/m)	3.7×10^{12}	1.1×10^{12}	3.7×10^{12}
Tangential stiffness, ks (Pa/m)	kn/10	kn/10	kn/10

2.3 *Structural elements*

The modeled concrete blocks are linked to each other thanks to structural elements. Piles elements represent the bars. These elements have an elastic compressive behavior, a limiting plastic moment and coupling parameters to define the interaction with the grid, here the zones of the blocks. The rockbolt option is used to describe an elastic-plastic behavior in tension, by choosing a tensile yield and a tensile failure strain. To model the plates, cable elements are added to laterally transfer forces and consequently limit displacements between blocks. The cable elements describe a perfectly plastic material that can yield in tension and compression, cannot resist a bending moment and can be grouted to interact with the grid.

A simplified model is proposed, in which the structural elements interact with the grid only through the node links where 6 degrees of freedom can be defined. Future developments will lead to improve the interaction model between the different structural elements by taking into account the gaps existing in the assembly. The coupling parameters and the grout parameters, respectively for the pile and the cable, may be used for this purpose.

2.4 *Other numerical considerations*

To keep realistic loading despite the relative important displacement of the impacted structure the large deformation option is activated. After the static stabilization of the wall under gravity, dynamic option is activated to solve the impact simulation.

In the future an additional numerical damping, probably Rayleigh damping, will be added to both limit the high frequency vibrations in materials and integrate dissipation phenomena existing for soft impact event that are not physically present in the numerical model.

3 FIRST IMPACT SIMULATION RESULTS

3.1 *Impact simulation method*

The impact is simulated in a realistic way, by throwing the projectile on the wall with an initial controlled velocity. The geometry of the projectile extremity impacting the structure is the same as that in the experiments. The mass of the projectile and its velocity magnitude and direction are chosen to simulate the impacts performed during the experimental tests. No additional numerical damping is considered in the presented simulations.

3.2 *Preliminary results for small scale experiments*

The impacted wall consists of 4 horizontal rows of 9 blocks, each having a length of 0.38m and a width and a height of 0.20m (Fig. 2). Piles and cables are created with a simplified rigid link model whereby displacements between the piles and the cable and between the structural elements and the blocks are blocked. A 690 J impact is simulated with a 35 kg projectile thrown, in the third row and in the middle of the wall, with a 6.3 m/s velocity.

The observed V shape of the wall post-impact is reproduced with a maximal displacement observed at the impact axis. The structure is deformed over a width of 4 blocks. The numerical model shows very similar deformation from the toe to the top, contrary to the experiments, where higher displacement is observed on the top row. This is attributed to the oversimplified model of the reinforcement elements, limiting the relative displacement of each block with its neighbors, in the same row. This is also considered to be the reason for the lower displacement observed with the numerical model than that measured after the impact experiments, with values of 8 and 16 cm respectively.

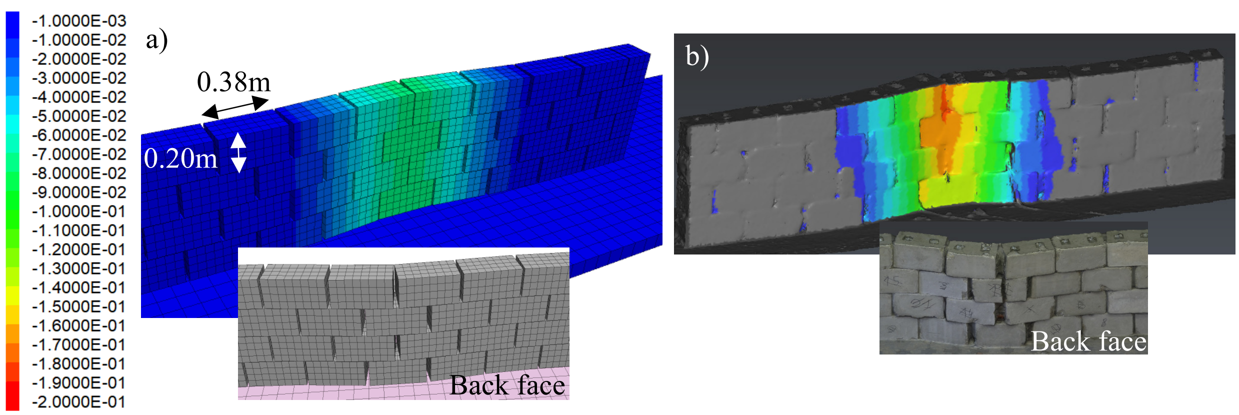

Figure 2. Horizontal displacement after 690 J impact on structure a) numerical simulation results b) experimental results.

4 CONCLUSIONS

An original numerical model involving interacting discrete blocks and structural elements is proposed for the design of an innovative protective structure against rockfall.

The *FLAC3D* developed model allows simulating a projectile impact on the structure taking into account the dynamic nature of the loading and the physical phenomena developed in the structure. The first impact simulation results show the capacity of the model in reproducing the global wall displacement. However, the results show the limits of the simplified model in describing the relative displacement and in reproducing the correct maximal displacement.

Ongoing developments aim at integrating in the model the complexity of the displacements and load transfer mechanisms between structural elements. Real impact tests results will be used to carry out the model calibration, by refining parameters, including damping. Simultaneously, a detailed analysis of the results will be conducted to verify the capacity of the model in reproducing known physical mechanisms and to prospect about energy dissipation phenomena.

In the long term, parametric studies will allow to optimize the structure design by identifying influent parameters associated with energy dissipation and result in an increase in the structure strength and a decrease in structure displacements after impact. In addition, the objective is to use the 3D numerical model as an engineering tool for design of rockfall protective structures or to develop tools or methods based on the simulation results.

REFERENCES

Breugnot, A., Lambert, S., Villard, P. & Gotteland, P. 2015. A discrete/continuous coupled approach for modeling impacts on cellular geostructures. Rock Mechanics and Rock Engineering, Doi: 10.1007/s00603-015-0886-8.

Itasca Consulting Group, Inc. 2017. *FLAC3D – Fast Lagrangian Analysis of Continua in 3-Dimensions, Ver. 6.0 User's Manual*. Itasca: Minneapolis.

Lambert, S., Heymann, A., Gotteland, P. & Nicot, F. 2014. Real-scale investigation of the kinematic response of a rockfall protection embankment. Nat. Hazards Earth Syst. Sci. 14, 1269–1281.

Peila, D., Oggeri, C. & Castiglia, C. 2007 Ground reinforced embankments for rockfall protection: design and evaluation of full-scale tests, Landslides, 4, 255–265.