Applied Numerical Modeling in Geomechanics – 2020 – Billaux, Hazzard, Nelson & Schöpfer (eds.) Paper: 10-03
 ISBN 978-0-9767577-5-7

PFC3D simulation of a compressed steel column

G. Forlati[1], A. Gajo[2] & L. Simeoni[2]
[1]*Metallurgica Ledrense Società Cooperativa, Trento, Italy*
[2]*Dipartimento di Ingegneria Civile, Ambientale e Meccanica, University of Trento, Trento, Italy*

1 INTRODUCTION

PFC3D (Itasca 2018) is a DEM software suited for geotechnical problems useful to describe the discrete nature of the soil. *PFC3D* is commonly used in granular and rock-mechanics problems in which the rock behavior is simulated with an assembly of particles. Rarely *PFC3D* is used to model a continuous material like structural elements. In usual practice, structural parts are modeled in *PFC3D* with continuous rigid elements which can translate and rotate and are not able to deform or bend. In order to have a structural part with bending resistance is necessary to model it with discontinuous elements. This problem of modeling is particularly visible when rigid gabions are studied. Gabions are formed by welded steel grids hooked to each other to form a box. Grids have their own bending resistance and they need to be treated as a continuous element. The steel box (Ledrosteel box 2019) is then filled with rock aggregates which are more suited for discrete modeling. The response of the steel can be simplified breaking the grids down into vertical columns and studying the behavior of a single steel wire column. In this paper the compression behavior of a grid steel wire is presented. The problem carries out the analysis presented in the Verification Problems of the *PFC* 6.0 interactive help menu.

2 DESIGN AND ANALYSIS

The steel wire studied had a length of 1 m and a diameter of 6 mm. It was placed in vertical direction and it was studied as a simply supported beam. In *PFC3D* a horizontal rigid plane was placed on the top of the column and then it was moved vertically downward to compress the wire once they got in contact. The load applied on the column was measured as 'contact force' between the moving wall and the top particle. The wire was modeled with a vertical row of 41 spherical particles of 25 mm of diameter in contact to each other (Fig. 1a). The first bottom particle was not allowed to move in the z- and y- direction and it formed a hinge. The last top particle was fixed in the y-direction while the particle was allowed to move in the vertical direction. The contact model assigned to the 40 contacts was the Linear Parallel Bond model because the bond carries both force and moment. The parallel bond normal and shear stiffness were calculated according to the verification problem 'Tip-loaded cantilever beam'. The normal and shear strengths were set to high values equal to the ultimate tensile strength of the steel. The normal and shear stiffness of the balls were calculated using the overall column stiffness. The last parameter to set was the radius multiplier $\bar{\lambda}$, which determines the dimension of the radius of the bond ($R_{bond} = \bar{\lambda} R_{particle}$). In a row made with identical particles the parallel bond contacts can be seen as a cylinder defined by its radius and length, making a 'column' between the particles. Thus, the parallel bond formed a cylindrical equivalent co-axial column to that made by the particles. The parallel bond contacts include axial bending and shear stiffness, so the bond area is an important parameter to set in order to have a rigid or flexible column. The radius multiplier was calibrated with the analytical bending solution of a simply supported column with a horizontal point load of 100 N acting in the middle section. The value of $\bar{\lambda}$ that ensured the same middle-section displacement between the numerical and analytical solution was 0.2337. The friction angle chosen for the analysis is the default value of 0.0. The parameters used for the contacts are reported in Table 1.

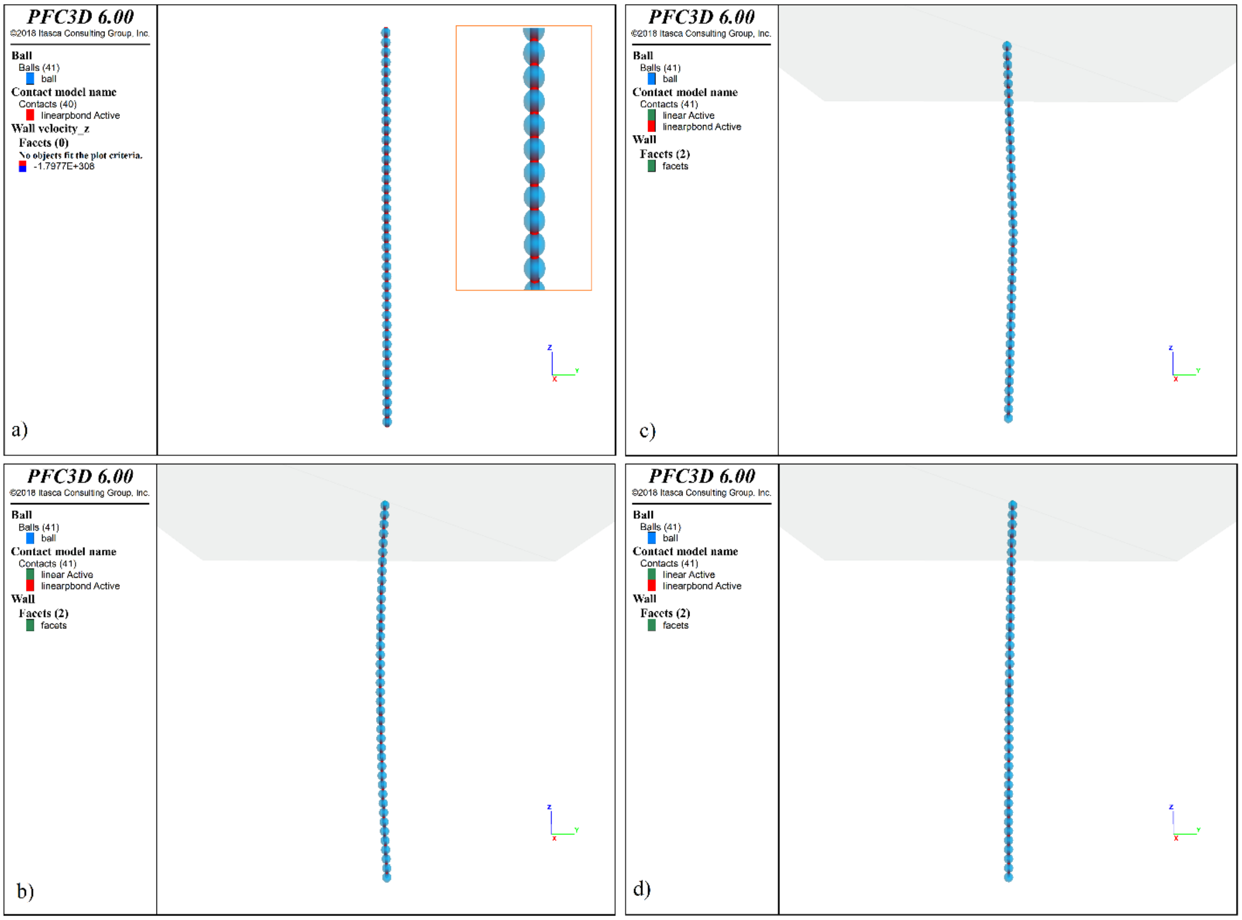

Figure 1. *PFC3D* model of a steel column a) with no initial flaw, b) with a parabolic shape, c) with a triangular shape, d) with a curvilinear shape.

Table 1. Particles and linear parallel bonds properties of the steel column calibrated in bending.

Parallel bond group			Linear group		
Normal stiffness (N/m^3)	$\bar{k}_n$	8.24×10^{12}	Normal stiffness (N/m)	k_n	4.50×10^8
Shear stiffness (N/m^3)	$\bar{k}_s$	3.23×10^{12}	Shear stiffness (N/m)	k_s	4.50×10^8
Tensile strength (Pa)	$\bar{\sigma}_c$	5.53×10^8			
Cohesion (Pa)	$\bar{c}$	5.53×10^8	Particle density (kg/m^3)	ρ	7850
Friction angle	$\bar{\phi}$	0.0			
Radius multiplier	$\bar{\lambda}$	0.2337			

3 RESULTS AND DISCUSSION

The calibrated column was tested in compression and in tension. The behavior in compression resulted different from that in tension. In tension the response was given only by the parallel bond group as the linear group of the contact model did not resist in tension. In compression, both the parallel bond and linear groups of the contact model were working, so the force applied to the column was divided by the two-parallel series of springs that represent the linear parallel bond contacts. The global stiffness ratio between compression and tension loading was equal to 2 as the compression displacements resulted half of those calculated in tension. An equal behavior in compression and in tension was obtained reducing to negligible values the interparticle contact stiffnesses. However, the reduction had the effect to increase the computational time. In the compression analysis of a single column the computational time increased 5 times when the particles stiffness was reduced to zero (from approximately 2 to 10 minutes of CPU time). A global stiffness ratio of 1.2-1.3 can be chosen as a compromise between an equal compression-tension model response and not excessive computational time. Thus, a particle normal and shear stiffnesses reduction from $4.50 \times 10^8\ N/m$ to $1.00 \times 10^8\ N/m$ was suggested to obtain a global stiffness ratio of 1.2.

In addition, a steel compressed column undergoes to buckling when the critical Euler load is reached. The value of the critical load can be calculated from theory and it is equal to 131 N for a simply-supported 1 m long column (Young's modulus E=210000 MPa, diameter Ø=6 mm, Fig. 2). Equation 1 reports the formula used to calculate the theoretical value of the critical Euler load.

$$P_{crit} = \frac{\pi^2 EI}{l_0^2} \quad (1)$$

where E is the Young's modulus, I the moment of inertia and l_0 is the effective column length, which is equal to the length of the column in the case of simply supported column.

PFC3D was not able to identify the critical Euler load when a perfect vertical column was used, and it gave an infinite resistance both in compression and in tension. For that an initial geometrical flaw was inserted into the column to understand if *PFC3D* identifies a critical load and if the column loses stability. The longitudinal axis was assumed to have a parabolic shape with an initial horizontal displacement in the middle section equal to 1.5 cm (Fig. 1b). The model lost stability with a large increase of the displacements at approximately 100 N and then broke at the contacts. Around 100 N small increments of the applied load caused the development of large displacement, meaning that the column was not able to sustain more load. This is visible in Figure 2, near the critical load the slope of the curve decrease comparing to the beginning of the compression test. The critical load recorded was smaller than the critical Euler load and the reduction was caused by the initial flaw. To study the dependence between the critical load and the type of flaw inserted, other two deformed columns were studied (Figs 1c & 1d). The longitudinal axis of the second column studied had a triangular shape with a middle section displacement of 1.5 cm. The third column had a curvilinear shape with a tip displacement of 1.5 cm. All the three modeled columns showed a critical load smaller than 131 N and lost stability around 100 N. Similar responses to that of the parabolic-shaped column were recorded during the compression analysis. The behavior in compression was studied also with a static analysis in which small force increments were applied to the column's tip and then *PFC3D* was run until the equilibrium was reached. In this manner the initial oscillations caused by the load application were reduced. The force-displacement graph obtained showed the loss of stability for buckling with the increase of displacements and agreed completely with the behavior obtained from the dynamic analysis (Fig. 2). The load reported in Figure 2 represents the vertical contact force measured on the horizontal wall and the displacements are the displacements of the horizontal wall. The static analysis was manually interrupted at the application of the 110 N load increment because the software could not reach the equilibrium. The displacement recorded during the application of the 110 N load increment was not reported in the graph. Static analysis had the advantage of decreasing the computational time.

The response of the model to different loading rate was then studied to check if there was any dependence of the applied velocity on the column's behavior. Applying a velocity of 0.01, 0.001 and 0.0005 m/s to the horizontal plane, the deformed column gave always the same behavior in terms of force-displacements (Fig. 2). However, a high velocity caused large oscillations of the contact force. Small velocities are suggested for a quasi-static analysis, but they have the negative effect of raising the computational time. For the studied problem, a velocity of 0.001 m/s was considered a good compromise between the quality of the results and the computational time. However, a static analysis should be considered as a valid alternative to study compression of columns, in particular when these are combined with other materials that can be affected by the initial oscillations of the force.

4 CONCLUSIONS

The study shows the ability of *PFC3D* to simulate structural continuous steel elements. Compression behavior of a column can be modeled using a row of spherical particles with linear parallel bond contacts. However, there are some features that need to be considered in order to obtain a reliable model. The column response can be calibrated in bending, compression and tension. Then buckling behavior can be checked. While for bending and tension the column gives the same results as the analytical solution, in compression the analytical and numerical solutions are different. The difference is caused by the bond contacts and it can be solved finding a balance between interparticle contact stiffness reduction and raising of computational time. Moreover, *PFC3D* is not able to analyze the buckling behavior when a perfect column is studied in compression. Flaws need to be inserted in order to obtain instability for smaller loads than the critical Euler load. The buckling analysis performed by *PFC3D* is independent from the loading rate. High and small loading rates give the same compression response at large displacements, while high loading rates cause oscillations in the particles at the beginning of the simulation. Small velocities and large computational times are required to minimize the initial oscillations, making a static analysis more appropriate for compression simulations.

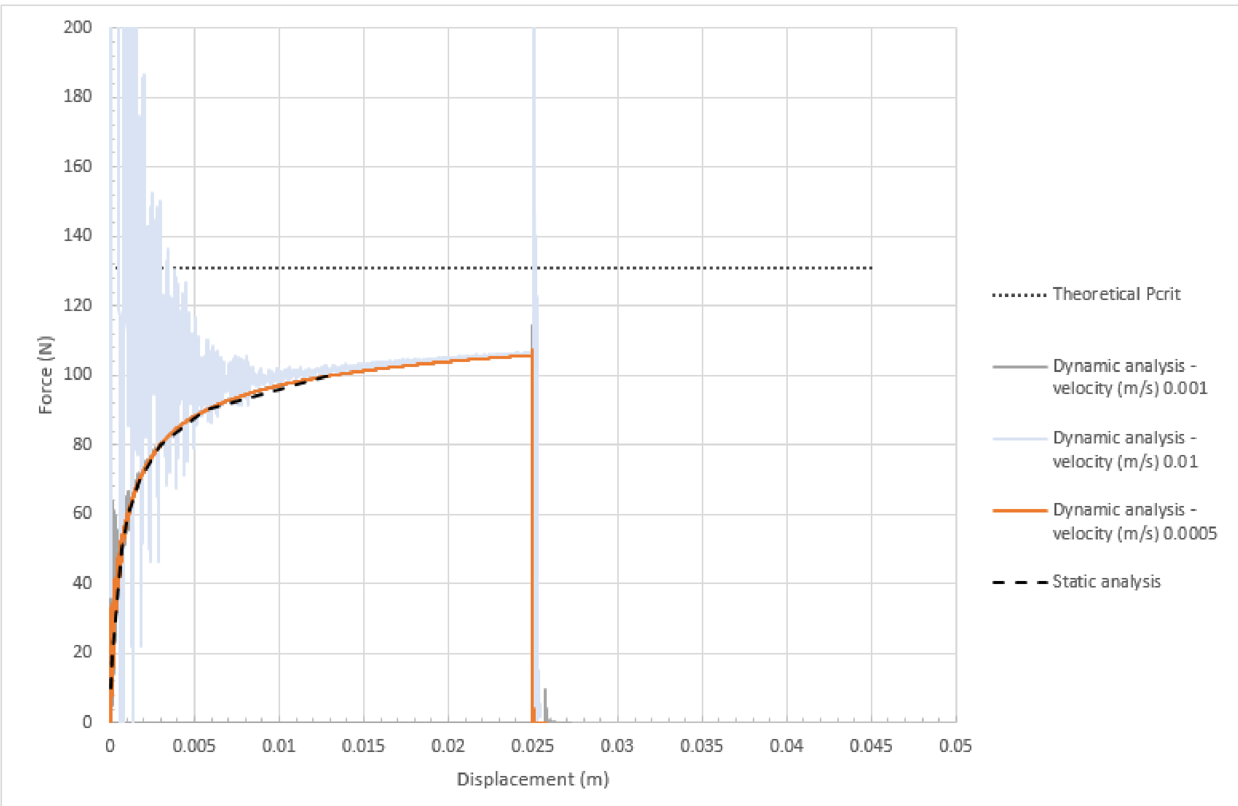

Figure 2. Dynamic and static compression responses of a triangular-shaped column.

REFERENCES

Itasca Consulting Group, Inc. 2018. *PFC3D – Particle Flow Code in 3 Dimensions, Ver. 6.0*. Minneapolis: Itasca.
Ledrosteel box website. 2019. https://www.ledrosteel-box.com/ (accessed 04 June 2019).