Preliminary finite element analyses on seismic resistant FREE from DAMage beam to column joints under impact loading

Marina D'Antimo[1, a *], Mariana Zimbru[2, b], Mario D'Aniello[2,c], Jean-François Demonceau [1,d], Jean-Pierre Jaspart [1,e] and Raffaele Landolfo[2,f]

[1] University of Liège, Faculty of Applied Sciences, ArGEnCo Department

Quartier Polytech 1, Allée de la découverte, 9, B52/3, 4000 Liège, Belgium

[2] of Naples 'Federico II', Department of Structures for Engineering and Architecture Via Forno Vecchio 36, Naples 80134, Italy

[a]m.dantimo@uliege.be, [b]mariana.zimbru@unina.it, [c]mdaniel@unina.it

[d]jfdemonceau@uliege.be, [e]Jean-Pierre.Jaspart@uliege.be,

[f]landolfo@unina.it

Keywords: FEM, robustness, steel joints, column loss, push-down tests, friction dampers.

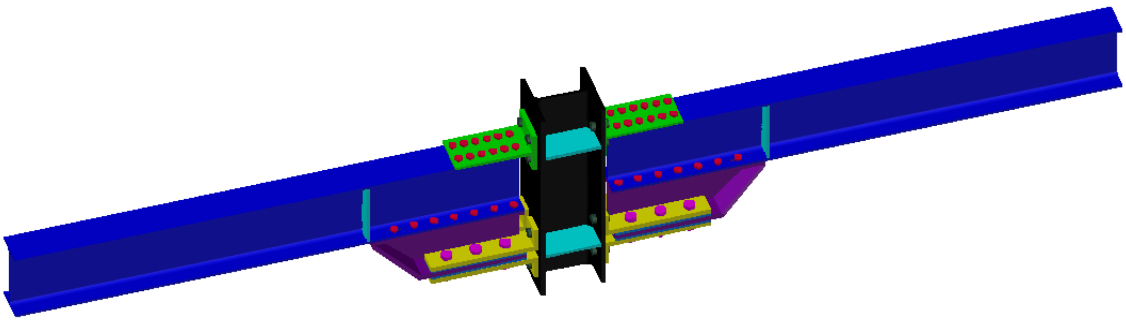
Abstract. Nowadays, the interest on structural robustness is increasing because of the recent terroristic attacks. Although a large number of research projects have been carried out in this field, limited design guidelines as well as code recommendations are nowadays available. Leading to the fact that the design for robustness is far from being current practice. Conversely, the design for natural hazards as the earthquake is a well-consolidated practice and modern codes implement effective and well-recognized design rules. Even though seismic design philosophy based on the concept of hierarchy of resistance enables structural robustness for conventional structural systems, this is not demonstrated for structures equipped with anti-seismic devices as well as innovative dissipative systems. Recently, the use of friction based dissipative joints has been proved to be a promising solution for seismically design steel moment resisting frames. However, the robustness and the resistance against impact loading of this type of joints is not yet investigated. With the aim to develop an experimental campaign based on impact tests, preliminary finite element analyses have been carried out to identify the main criticisms and to drive the rational design of the joint specimens. With this regard, in the present paper, the results of a numerical parametric study on the preliminary push-down test are presented and discussed.

Introduction

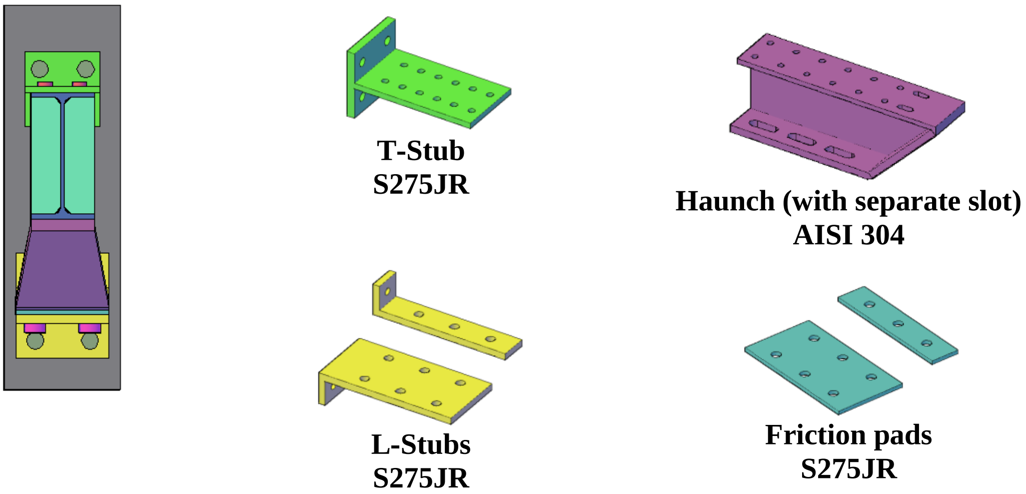
In the framework of the RFCS FREEDAM (acronyms for Free From Damage connections [1]) project, novel dissipative beam to column connections for seismic moment resisting frames (MRFs) are under development. The main peculiarity of this connection relies on the dissipative mechanism that is based on friction developing between a set of bolted plates covered by special high friction layers. In such a way, these connection can withstand severe and frequent seismic events without suffering of almost any damage, which is cost-effective if compared with traditional connections where plastic deformation are expected to occur. Hence, it is trivial to recognize the large benefit of FREEDAM connections in seismic application. Conversely, despite the growing interest in the subject [2]–[8], the performance under unconventional actions (e.g. impact, column loss, blast, etc.) is far from being clarified. Therefore, the robustness of these connections due to impact or blast is currently under investigation within the framework of FREEDAM project. In this paper the preliminary outcomes of this study are presented and discussed. Prior to the robustness investigations, a calibration of a FEM model on the basis of the available quasi-static response of the joint is essential to have a valid tool for the investigation of the dynamic response.

Pushdown tests on FREEDAM connection

The tested specimen is an internal beam-to-column joints (see Fig. 1a). IPE 200 are adopted for beams and HEB200 for the column and S275 steel grade is assumed for all members. The bottom flange of the beam is connected to a haunch bolted to the column by means of cleat angles (see Fig. 1b). Between the haunch and the angles there are friction pads on both sides, with a special coating, which ensures the proper friction behavior of the device (see Fig. 1c). The whole package is fastened together with M20 bolts class 10.9 HV, which are free to move in slotted holes, calibrated according to the reference design earthquake [1].



a)



b) c)

Fig. 1. a) Tested layout ; b) Lateral view; c) Joint details, from [9]

The load, applied downwards with an hydraulic jack on the top of the column, leads to sagging bending moment in the joint. In Fig. 2a all the measurements are indicated in their location. The letter D indicates the transducers to measure the following displacements: at right and left support (i.e. D-RG and D-LF) and the vertical displacement of the column (D-V); the relative displacements between the external plate and the haunch on both sides (DR-1and DR-2) and the relative displacements between the two haunch front and back sides (DR-F and DR-B). The letter I marks the inclinometers, as follows: IM-01 and IM-02 used to measure the relative rotation of the beams and IM-03 for the central column rotation.

The performed test shows the first slippage of the device occurring at 100 kN, and a second slippage of the upper T-Stub at about 150 kN (see the response curve in Fig. 2c). After the first slippage, new components are activated in the connection (bolt in shear and plate components in bearing), which lead to hardening behaviour up to the occurrence of the instability of the beam.

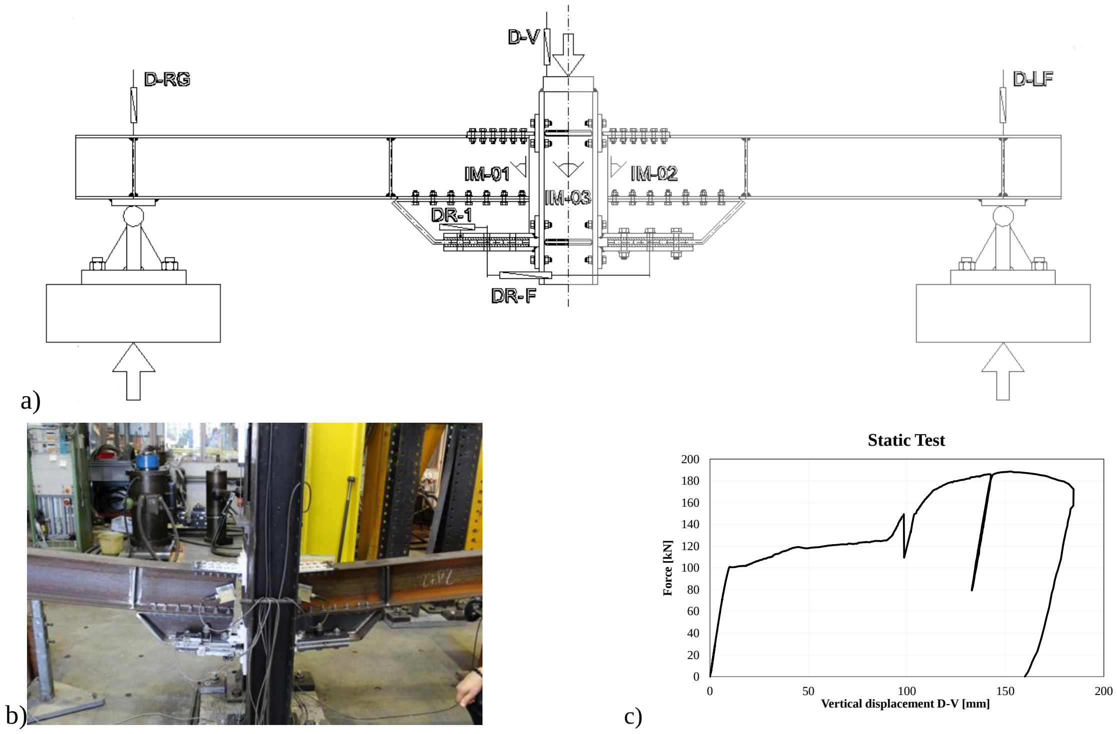


Fig. 2. Static test results: a) specimen at the end of the test; b)Force displacement curve at the central column;

FE Analysis

The finite element (FE) model is developed using the commercial software Abaqus v6.14. Dynamic implicit quasi-static analysis is performed to simulate the behavior of the connection. In order to reduce the size of the model, only half geometry is modelled and the boundary conditions are assumed to account for the symmetry. The extremity of the beam is supported by a vertical roller, hence, the axial translation is unrestrained.

C3D8R 8-node linear brick with reduced integration is used for almost all the elements except for the beams that are meshed using C3D8 elements (i.e. with full integration and first order approximation) in order to mimic the buckling phenomena more effectively ([10], [11]). The adopted mesh is shown in Fig. 3. The material properties are implemented in the model by means of multilinear curve based on the true stress-strain curves obtained from coupon tensile tests performed at the University of Coimbra. The bolts are modelled using the nominal diameter and scaling the material nominal stress as explained in [12]–[16]. The bolt preload is applied to the middle section of the bolt shank, using the option “Bolt Force” available in Abaqus/CAE and the magnitude was calculated with the prescription given by Eurocode 3 Part 1-8. All the interactions are modelled using surface-to-surface contact formulation. Normal “hard contact” and tangential behavior are defined for all the interactions, for steel-to-steel interaction a friction coefficient of 0.3 is considered, while the friction coefficient of the device is defined on the basis of the experimental results obtained within [1].

The geometric imperfections are introduced into the “ideal” model by means of linear superposition of buckling Eigen modes. The magnitude of the initial imperfection is established according to [17]. Based on the presented assumptions, the analysis is performed in two steps, namely in the first one the preload is applied to the bolts and in the second one a vertical displacement is imposed to the middle of the column (at a defined Reference Point).

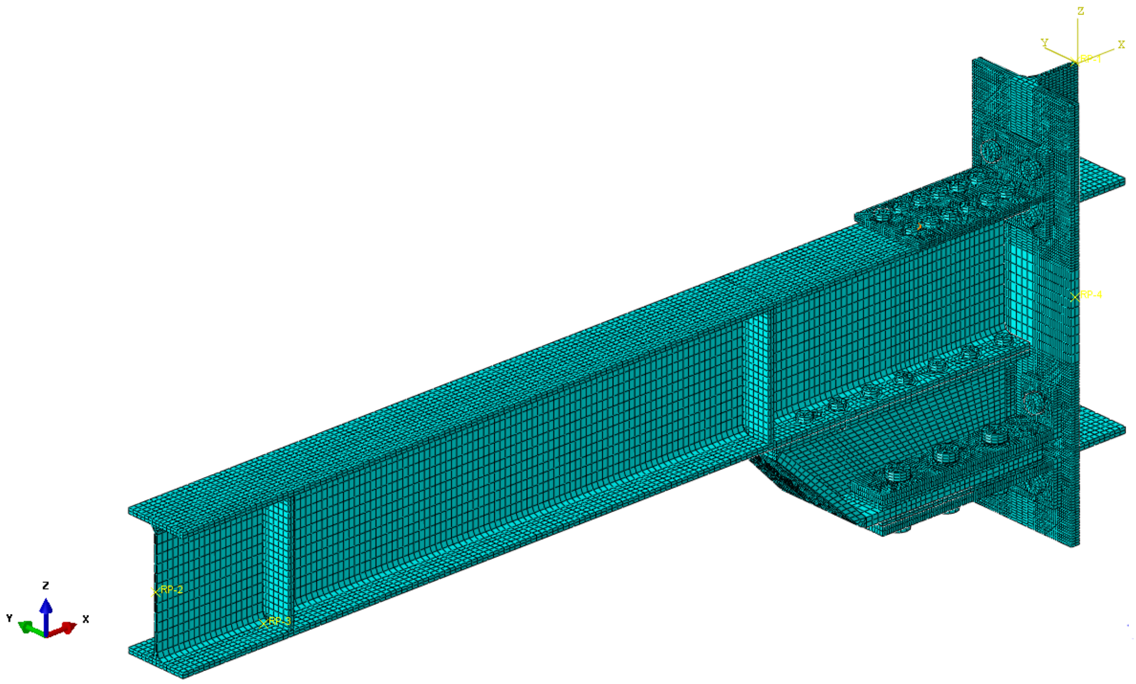


Fig. 3. Meshed model

Numerical results and comparison

The results of the FE analysis in term of moment rotation curve are given in Fig. 4 and compared with the experimental results. The friction coefficient have been ranged between the upper and the lower bound values [1] to identify the actual value.

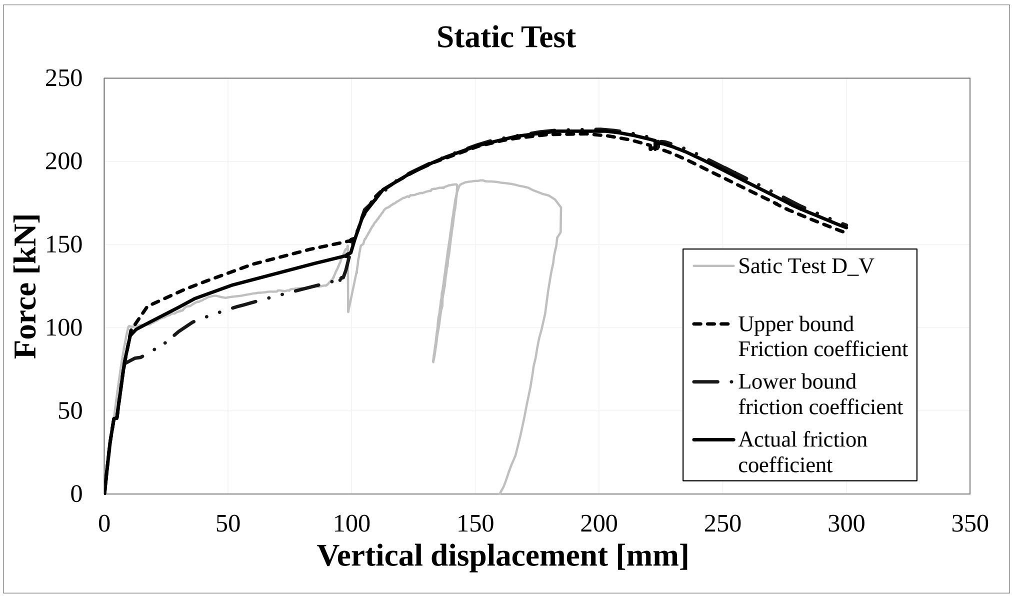


Fig. 4. Comparison between FE analysis and experimental test.

The value of the resistance obtained with the FE analysis is 15% higher than the experimental one. This overestimation can be linked to the definition of the restrain in the FEM, which is modelled as a perfect roller. In the reality the lower flange of the beam can exhibit an unilateral displacement. Thus, two limit conditions were considered in order to assess the influence of the restrain definition on the development of the instability, the real behaviour can be seen somewhere in between the two limit conditions (in Fig. 5). With the second limit condition, the resistance is slightly underestimate (8%).

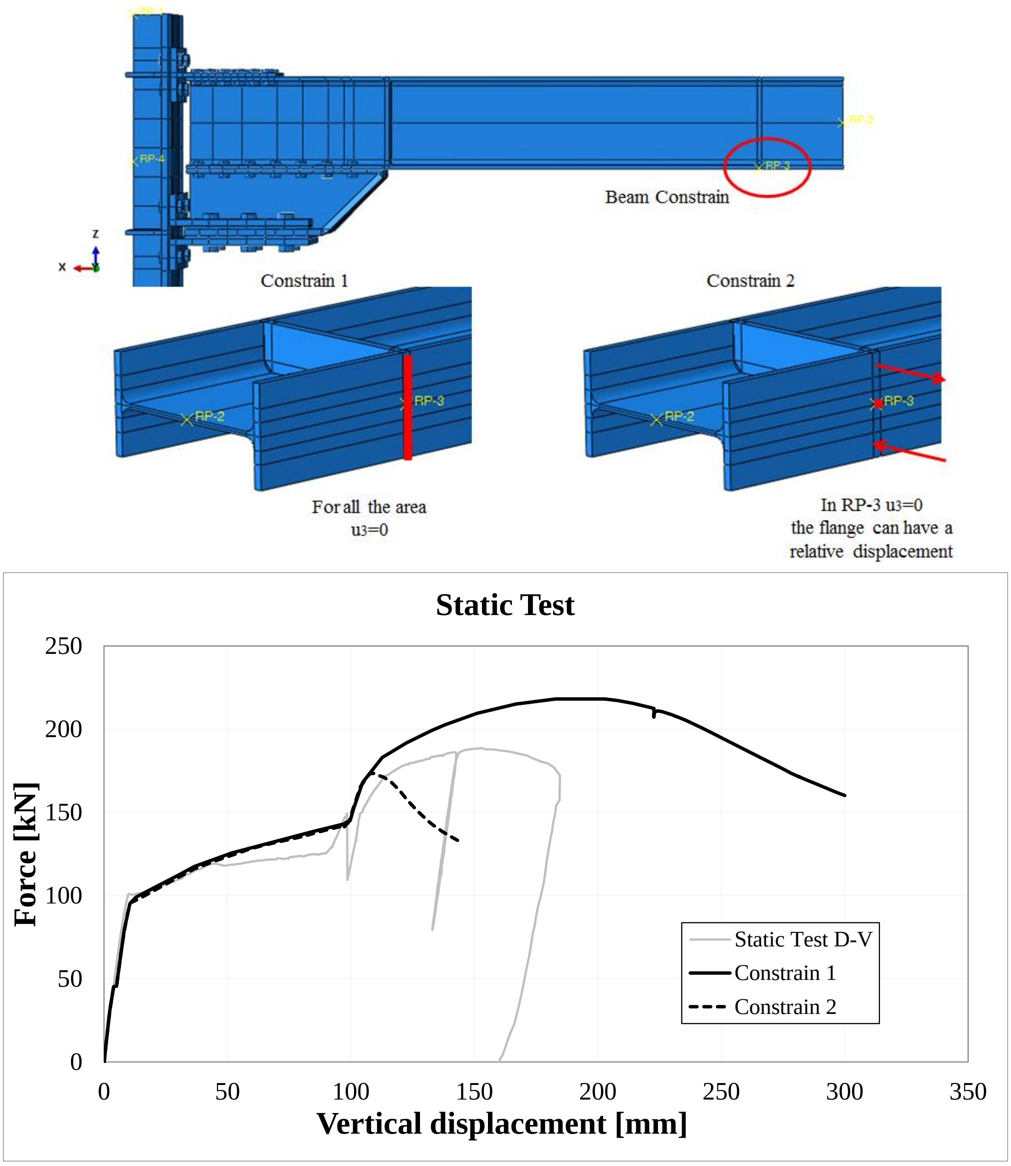


Fig. 5. Comparison of two limit constrain conditions.

In Fig. 6 the deformed shape of experimental test and finite element prediction are compared and a good agreement can be found, however. Fig. 7a-d summarizes the stress evolution in the tested connection for different value of the vertical displacement.

In the same figure the distribution of the plastic damage in the elements (PEEQ, cumulative plastic deformation) at the end of the test is given. The largest concentration of the plastic deformation are in the upper beam flange, due to the occurrence of local instability, and in the lower beam flange where the opening of the connection with the haunch almost caused the bolt failure.

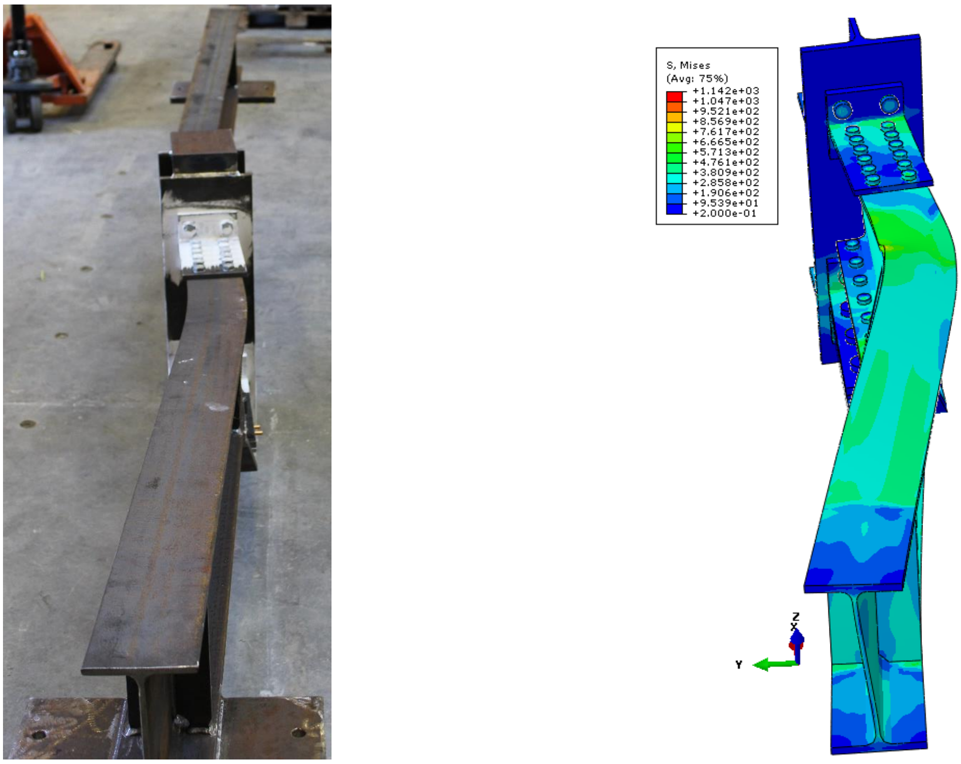
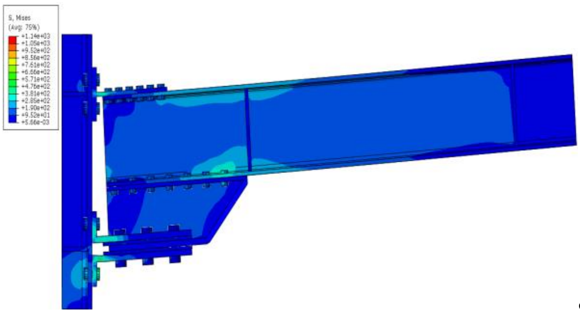
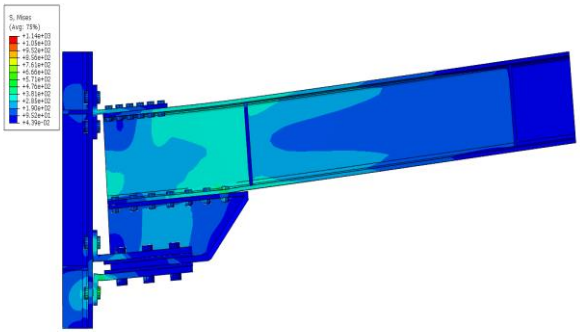


Fig. 6. System built for the dropping mass



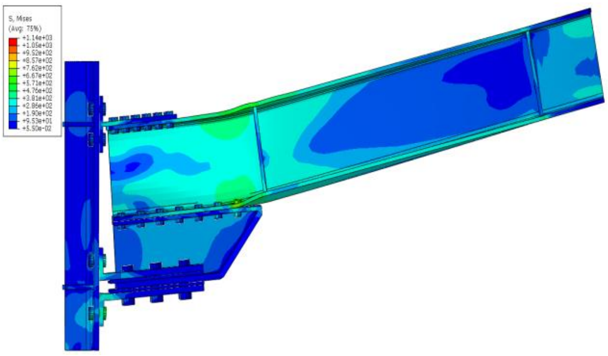
a)

d= 85mm



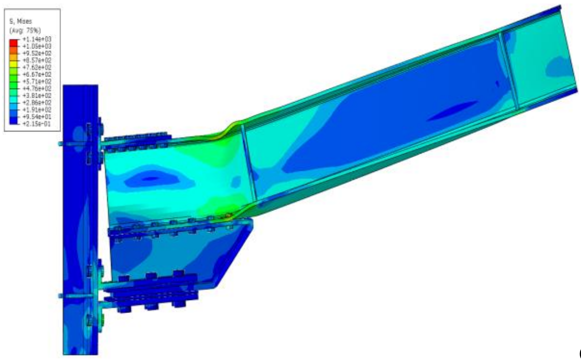
b)

d=130 mm



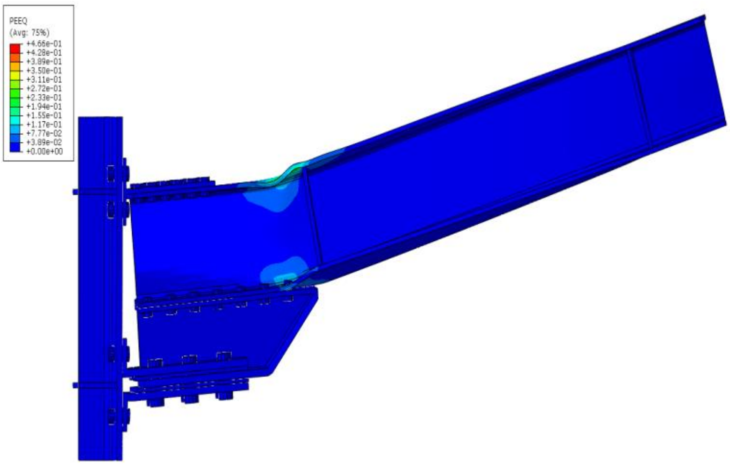
c)

d=235mm



d)

d= 300mm



e)

d= 300mm

Fig. 7. a)-d) stress distribution at different level of displacement; d) Cumulative plastic damage at the end of the test;

Summary

The paper describes the numerical modelling of the push down test performed in the framework of the FREEDAM project. The aim of the work is to characterize the behavior of the proposed connection in robustness application, especially under the column loss scenario. To this aim a preliminary pushdown test is performed in order to calibrate a FE model to perform further parametrical analysis. The FE simulation satisfactory matches the experimental response. In addition, the numerical analysis shows that the damage pattern move from the device to the connected beams, which play a key role to arrest the column loss .

References

[1] n° RFSR-CT-2015-00022, "Research Fund for Coal and Steel (RFCS) research programme under grant agreement."

[2] E. Grimsmo, C. A.H., A. Aalberg, and M. Langseth, "Beam-to-column joints subjected to impact loading," no. September, 2015.

[3] J.-F. Demonceau, L. Comeliau, L. Hoang Van, and J.-P. Jaspart, "How can a steel structure survive to impact loading? Numerical and analytical investigations," *Open Civ. Eng. J.*, vol. 11, pp. 434–452, 2017.

[4] C. C. Segura, L. Hamra, M. D 'Antimo, J.-F. F. Demonceau, and M. Feldmann, "Determination of Loading Scenarios on Buildings Due to Column Damage," *Structures*, vol. 12, pp. 1–12, 2017.

[5] D. Cassiano, M. D'Aniello, and C. Rebelo, "Parametric finite element analyses on flush end-plate joints under column removal," *J. Constr. Steel Res.*, vol. 137, pp. 77–92, 2017.

[6] J.-F. Demonceau, H. Vanvinckenroyea, M. D'Antimo, V. Denoel, and J.-P. Jaspart, "Beam-to-column joints, column bases and joint components under impact loading," in *EUROSTEEL 2017, September 13–15*, 2017.

[7] C. Huvelle, V.-L. Hoang, J.-P. Jaspart, and J.-F. Demonceau, "Complete analytical procedure to assess the response of a frame submitted to a column loss," *Eng. Struct.*, vol. 86, no. March, pp. 33–42, 2015.

[8] J.-P. Jaspart, J.-F. Demonceau, and L. Comeliau, "Robustness of steel and composite buildings suffering the dynamic loss of a column," *7th Natl. Conf. steel Struct. Proc.*, pp. 71–86, 2011.

[9] M. D'Antimo, M. Latour, G. Rizzano, J.-F. Demonceau, and J.-P. Jaspart, "Preliminary Study on beam-to-column joints under impact loading," *Open Constr. Build. Technol. J.*, no.

Special issue, 2017.
[10] M. D'Antimo, J. F. F. Demonceau, J. P. P. Jaspart, M. Latour, G. Rizzano, M. D 'antimo, J. F. F. Demonceau, J. P. P. Jaspart, M. Latour, and G. Rizzano, "Experimental and theoretical analysis of shear bolted connections for tubular structures," *J. Constr. Steel Res.*, vol. 138, pp. 264–282, 2017.
[11] O. S. Bursi and J.-P. Jaspart, "Calibration of a finite element model for isolated bolted end-plate steel connections," *J. Constr. Steel Res.*, vol. 44, no. 3, pp. 225–262, Dec. 1997.
[12] M. D'Aniello, D. Cassiano, and R. Landolfo, "Monotonic and cyclic inelastic tensile response of European preloadable gr10.9 bolt assemblies," *J. Constr. Steel Res.*, vol. 124, pp. 77–90, 2016.
[13] M. D'Aniello, R. Tartaglia, S. Costanzo, and R. Landolfo, "Seismic design of extended stiffened end-plate joints in the framework of Eurocodes," *J. Constr. Steel Res.*, vol. 128, pp. 512–527, 2017.
[14] M. D. Aniello, M. Zimbru, M. Latour, and A. Francavilla, "Development and validation of design criteria for free from damage steel joints," vol. 1, no. 2, 2017.
[15] M. D. Aniello, M. Zimbru, R. Landolfo, M. Latour, G. Rizzano, and V. Piluso, "FINITE ELEMENT ANALYSES ON FREE FROM DAMAGE SEISMIC RESISTING BEAM-TO-COLUMN JOINTS," no. June, pp. 15–17, 2017.
[16] M. D'Aniello, D. Cassiano, and R. Landolfo, "Simplified criteria for finite element modelling of European preloadable bolts," *Steel Compos. Struct.*, vol. 24, no. 6, pp. 643–658, 2017.
[17] L. S. Da Silva, R. Simões, and H. Gervásio, *Design of Steel Structures - 2nd Edition*. ECCS, 2016.