DETAILED FINITE ELEMENT ANALYSIS OF STEEL MOMENT FRAMES

Apostolis A. Vrakas, M.Sc.
Aristidis G. Papachristidis, Ph.D.
Manolis Papadrakakis, Professor
National Technical University of Athens
Zografou Campus, Athens 15780, Greece
apostolis.vrakas@gmail.com

1. ABSTRACT

The objective of this paper is the nonlinear detailed finite element analysis of steel moment resisting frames with extended end-plate bolted beam-to-column joints. Firstly, the simulation of the joints is performed, using structural (beam and shell) and three-dimensional continuum (eight-node hexahedral solid) elements. Material as well as geometric nonlinearities with contact between the appropriate components of the connections are taken into account. The moment-rotation (M-φ) response of characteristic joints, subjected to static loads, is calculated and compared with experimental results and EC3 predictions for the validation of the corresponding numerical models. Then, multistory steel frames are studied, using hybrid finite element models with beam-column elements for the structural members and detailed simulation of the joints assuming proper compatibility constraints at the interface sections. Pushover analyses are performed demonstrating the effect of joints on the overall behavior of the structure. Finally, stiffening of the joints with supplementary web plates is performed in order to examine the influence of the panel zone deformations.

2. INTRODUCTION

Bolted beam-to-column joints with extended end-plates are used widely in steel structures. They form moment resistant connections between steel members, but their behavior can be either rigid or semi-rigid depending on the stiffness and strength of their components. The consideration of flexible joints corresponds to a more realistic simulation of the frame behavior leading to more reliable solutions. However, the large number of variables required for the simulation of the connection geometry makes the task of taking into consideration the semi-rigid behavior of joints into the frame design a complicated process. Additionally, structural joints may exhibit nonlinear behavior such as localized elastoplastic deformations, unilateral contact and slip phenomena. The behavior of steel joints has been the subject of both experimental [7-9] and numerical [3-6] studies by a number of researchers.

This paper first presents a finite element study of semi-rigid joints subjected to static loading. Stiffness, moment resistance and rotation capacity derived from the calculation of moment-rotation (M-φ) curves are compared with experimental results by Coelho et al. [7] and Eurocode 3 recommendations [2]. Then, multistory steel frames with detailed modeling of their joints are examined, while parametric studies are performed demonstrating the effect of geometric characteristics variations of joint components on the connection behavior and consequently on the overall response of the frames. The finite element discretization of joint and frame models is produced automatically from their geometric description via appropriate code that has been developed. Abaqus/Standard software is used for the numerical analyses of this paper [1].

3. FINITE ELEMENT MODELING OF STEEL JOINTS

3.1 Simulation with shell elements

Two different element types are used for modeling the end-plate bolted beam-to-column joint (Fig. 1). Plane components of the joint (beam/column flanges and web, end-plate, transverse web stiffeners) are modeled with the S4 quadrilateral shell element of appropriate thickness, while the bolts are modeled with beam elements of circular section. The interaction between the column flange and the end-plate is considered through surface-based contact simulation with element-based surface definition, which enables contact between independent meshes without node compatibility.

3.2 Simulation with continuum elements

Three-dimensional eight-node hexahedral solid elements are used for the detailed simulation of the extended end-plate bolted beam-to-column joint (Fig. 1). Apart from the finite element discretization difficulties using three-dimensional solid elements, there are many complexities related to the contacts between the different components of a beam-to-column bolted joint. Particularly, there are five interactions that should be considered: (a) column flange with end-plate; (b) column flange with bolt nut; (c) end-plate with bolt head; (d) column flange hole with bolt shank; and (e) end-plate hole with bolt shank. All of them are modeled using surface-to-surface contacts. Surfaces are defined through the appropriate faces of the hexahedral solids. More specifically, for cases (a), (d) and (e) small sliding contact formulation is considered with a softened contact relationship. The slope of the linear pressure-overclosure curve is taken equal to 10^5. The tangential behavior of the interfaces is modeled through the basic isotropic Coulomb friction model with a constant coefficient μ equal to 0.30. For the other cases tied contact simulation is considered, where each node on the slave surface has the same displacement with the corresponding contact point on the master surface.

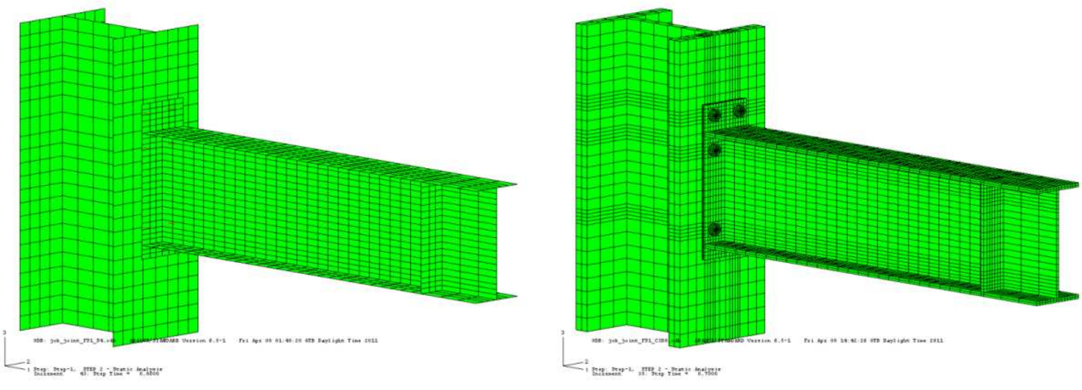


Fig. 1: Joint simulation with quadrilateral shell finite elements and 8-node hexahedral solid finite elements respectively

3.3 Numerical results

An experimental investigation of eight statically loaded extended end-plate moment connections was undertaken at the Delft University of Technology by Coelho et al. [7] to provide insight into the behavior of this type of joints up to collapse. The parameters investigated were the end-plate thickness and steel grade. Geometric and mechanical characteristics, boundary conditions and loading procedures are considered the same as in the experiments. The finite element analysis procedure is based on the incremental Newton-Raphson technique, while material and geometric nonlinearities are taken into account via the von Mises isotropic plasticity model and large displacement consideration.

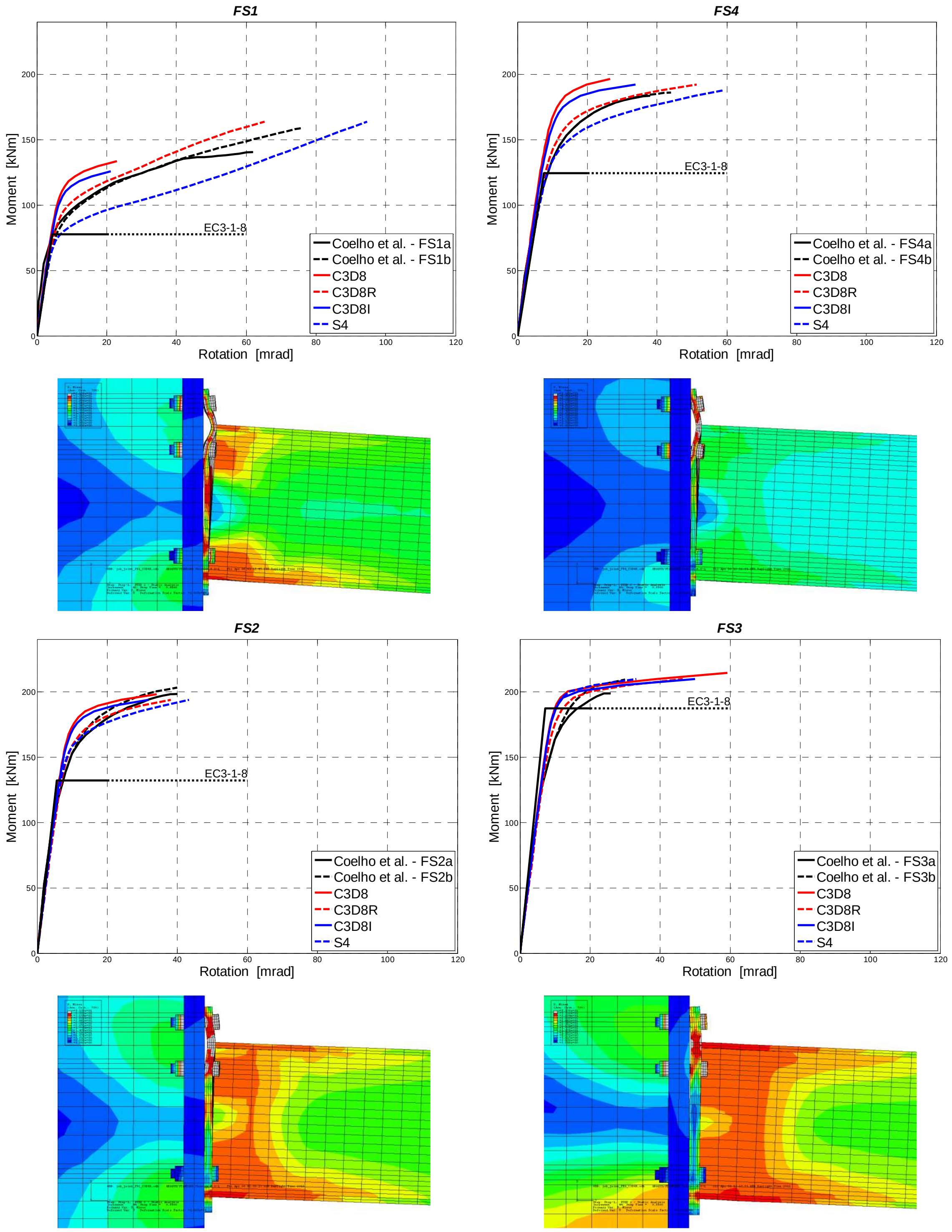


Fig. 2: Numerical vs. experimental results

Moment-rotation (M-φ) curves are presented in Fig. 2 and compared with the experimental results [7] and Eurocode 3 suggestions [2]. Moreover, for each test, horizontal deformation of the end-plate is observed. The first three models are plotted in stress scale 0-35.5 kN/cm^2 and the last in 0-69.0 kN/cm^2 according to each end-plate yield stress. Four finite element models are examined, one with shell elements and three with continuum elements. All of them predict accurately the elastic stiffness of the joints. Bending of end-plate is the main reason for differences between these simulations in the elastoplastic region. For thin end-plates, which are dominated by bending (FS1, FS4), the use of solid elements with reduced integration (C3D8R) performs better. This simulation predicts almost exactly the main joint behavioral characteristics: stiffness, resistance and rotation capacity. The normal brick element (C3D8) overestimates the elastic behavior of the joint, while the enhanced with incompatible modes (C3D8I) gives better results but remains stiff for this type of problems underestimating their high rotation capacity. Joint modeling with shell elements has a drawback. Bolt forces are applied locally to the nodes that connect the column flange with the end-plate and as a result there is local concentration of stresses, especially for very thin end-plates and/or column flanges. As the thickness of end-plate increases (FS2, FS3) the simulations tend to coincide. Brick elements with reduced integration remain the more flexible, while bricks with incompatible modes give also reliable results. Simulation with shell elements behaves very satisfactory and can be used for such type of problems.

4. FINITE ELEMENT MODELING OF STEEL FRAMES

4.1 Numerical modeling

Hybrid models have been developed with beam-column elements for the structural members and detailed modeling of the joints according to section 3.1 with shell elements. Proper compatibility constraints are assumed at the nodes of the interfaces, while the length of the detailed joint part (beam/column) is taken as 0.10 of the corresponding total member length. There are also some results from simple models with beam-column elements for the members and no specific modeling of the joints (regarded as rigid). Nonlinear static analyses under horizontal loads taking into consideration P-delta effects have performed. The pushover analyses distribution of the horizontal loads is considered triangular along the height of the structure, while the base joints are fixed. Two different frame models are examined with the connections as shown below and steel grade *S235*.

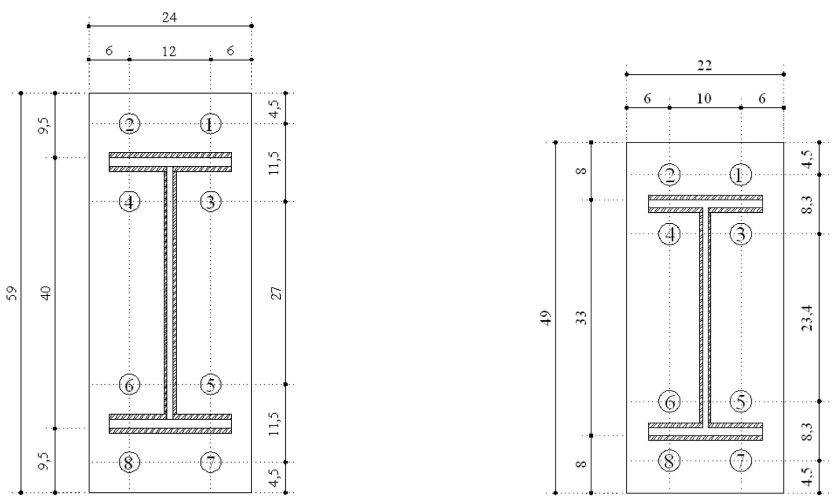


Fig. 3: Connections of Frame A & B respectively

4.2 Frame A

The first steel frame has four openings (4 x 7m = 28m) and three stories (3 x 4m = 12m) as shown in Fig. 4. Columns have a section profile HEB320 and beams IPE400. Details of the beam-to-column connections are shown in Fig. 3. Each beam is assumed to have 20 times higher weight to account for the floor mass (~ 36000kg), while each column carries its own weight. The first two natural periods of the frame are 0.60 and 0.20 seconds respectively.

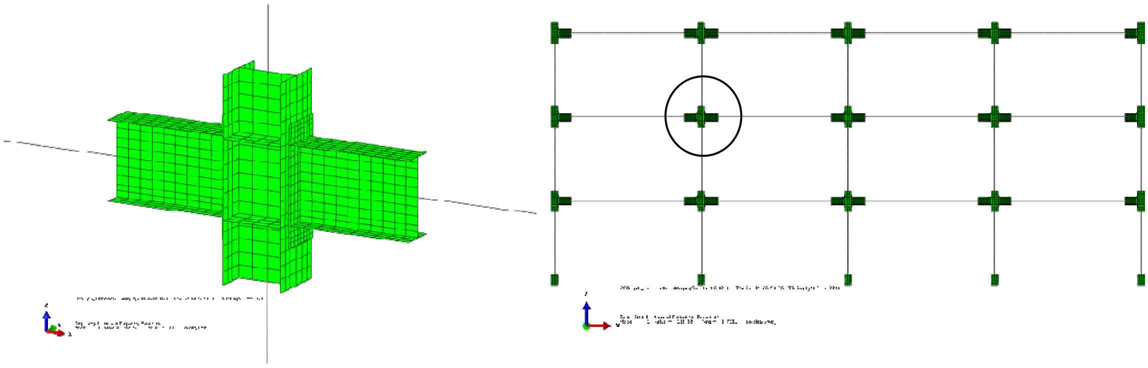
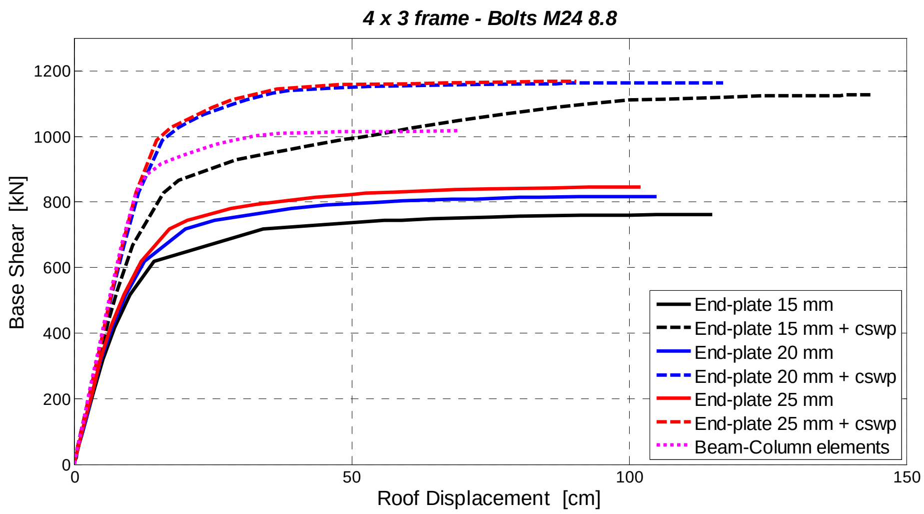
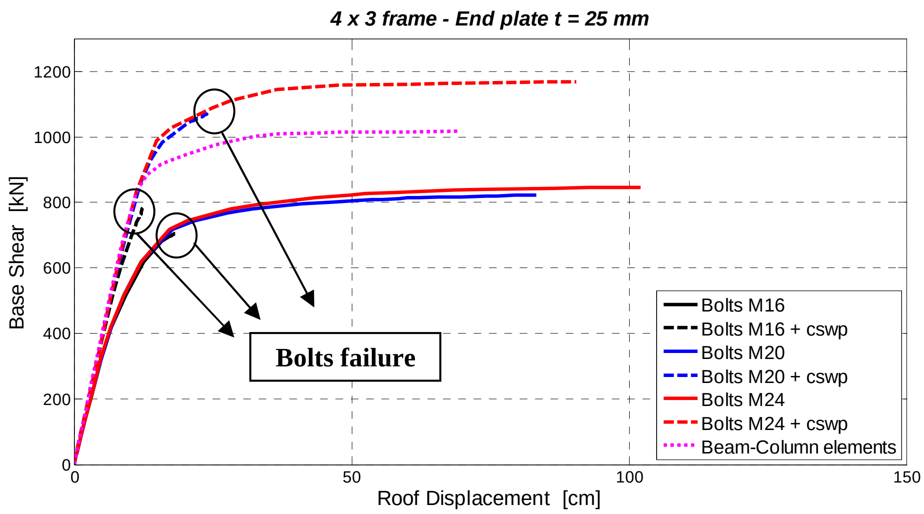


Fig. 4: Finite element model of Frame A





Figs. 5, 6: Frame A – Numerical results

As can be seen from Figs. 5, 6, adding column supplementary web plates (*cswp*, with thickness $t'_{wc} = 3 \times t_{wc}$) to the joints is very effective, decreasing the shear deformation of the column web panel zones and improving the overall behavior of the frame.

4.3 Frame B

The second frame has three openings (3 x 6m = 18m) and six stories (6 x 4m = 24m) as shown in Fig. 7. Columns have a section profile HEA400 and beams IPE330. Details of the beam-to-column connections are shown in Fig. 3. Each beam is assumed to have 10 times higher weight to account for the floor mass (~ 8500kg), while each column carries its own weight. The first two natural periods of the frame are 0.80 and 0.25 seconds respectively.

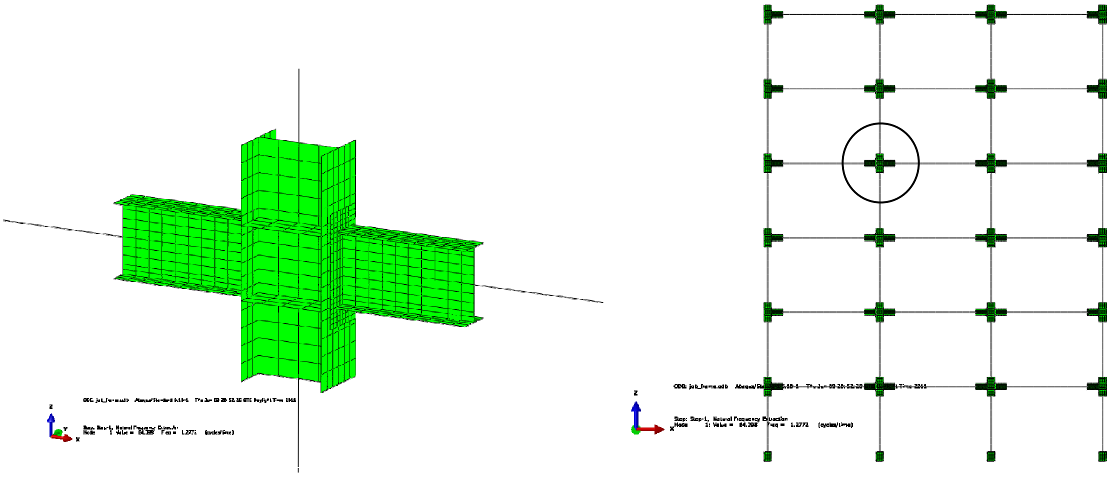
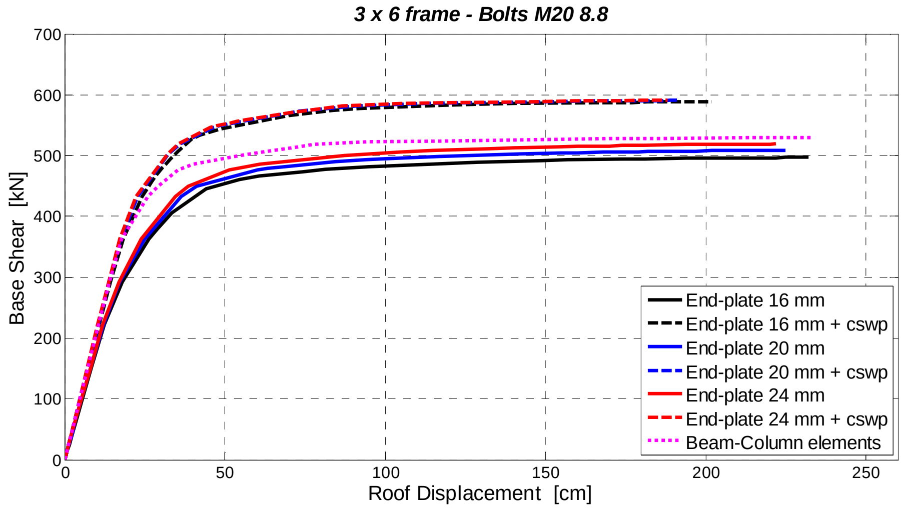
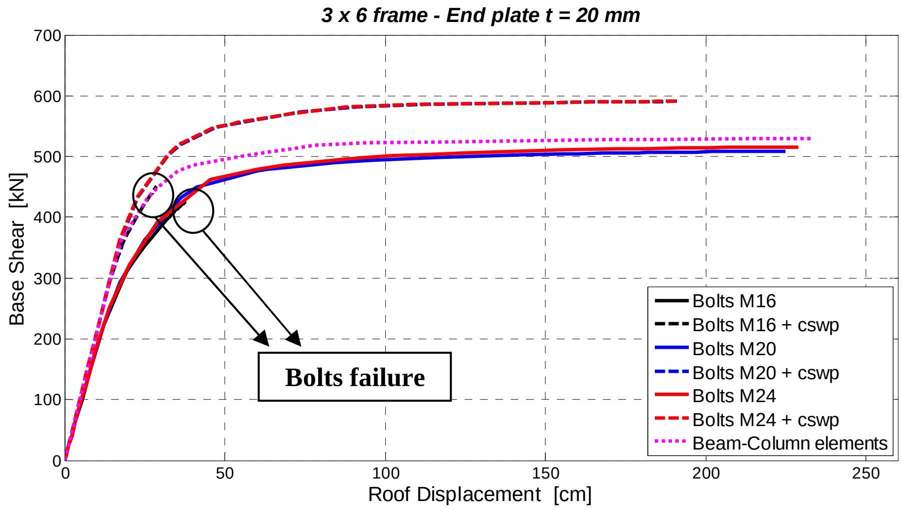


Fig. 7: Finite element model of Frame B





Figs. 8, 9: Frame B – Numerical results

5. CONCLUSIONS

- Experimental results can be reproduced accurately by detailed finite element models considering material and geometric nonlinearities including appropriate contacts of bolted extended end-plate beam-to-column joints under static loads.
- Hybrid frame models, where the joints are modeled with shell elements and the connecting structural elements with beam-column elements, assuming appropriate kinematic constraints at the interface sections, can be used for the nonlinear static analysis of moment resisting frames capturing all local phenomena that affect the behavior of the joints at reduced computational times.
- The shear deformation of the column web panel zone of a joint is of equal importance to the connection rotational deformation and can dominate the inelastic response of a moment frame. Hence, joint modeling is needed to realistically predict the overall frame performance.

6. REFERENCES

[1] ABAQUS version 6.10, *Analysis User's Manual*. 2010

[2] European Committee for Standardization (CEN), *EN 1993-1-8 – Eurocode 3: Design of Steel Structures –Part 1.8: Design of Joints.* 2005

[3] M.R. Bahaari, A.N. Sherbourne, Computer modelling of an extended end-plate bolted connection. *Computers and Structures*, **52**(5), 879-893, 1994

[4] O.S. Bursi, J.P. Jaspart, Calibration of a finite element model for isolated bolted end-plate steel connections. *Journal of Constructional Steel Research*, **44**(3), 225-262, 1997

[5] O.S. Bursi, J.P. Jaspart, Basic issues in the finite element simulation of extended end plate connections. *Computers and Structures*, **69**, 361-382, 1998

[6] Anant R. Kukreti, Feng-Feng Zhou, Eight-bolt endplate connection and its influence on frame behavior. *Engineering Structures*, **28**, 1483-1493, 2006

[7] Ana M. Girao Coelho, Frans S. K. Bijlaard, Luis Simoes da Silva, Experimental assessment of the ductility of extended end plate connections. *Engineering Structures*, **26**, 1185-1206, 2004

[8] Ana M. Girao Coelho, Frans S.K. Bijlaard, Experimental behaviour of high strength steel end-plate connections. *Journal of Constructional Steel Research*, **63**, 1228-1240, 2007

[9] Yongjiu Shi, Gang Shi, Yuanqing Wang, Experimental and theoretical analysis of the moment-rotation behaviour of stiffened extended end-plate connections. *Journal of Constructional Steel Research*, **63**, 1279-1293, 2007

[10] Akshay Gupta, Helmut Krawinkler, Seismic demands for performance evaluation of steel moment resisting frame structures. *The John A. Blume Earthquake Engineering Center*, Report No. 132, 1999

[11] Kee Dong Kim, Michael D. Engelhardt, Monotonic and cyclic loading models for panel zones in steel moment frames. *Journal of Constructional Steel Research*, **58**, 605-635, 2002

[12] A.A. Vrakas & M. Papadrakakis, A study of the influence of the rigidity of joints on the dynamic response of steel structures. *3rd ECCOMAS Thematic Conference on Computational Methods in Structural Dynamics and Earthquake Engineering (COMPDYN 2011)*, Corfu, Greece, 25-28 May 2011

ΑΝΑΛΥΣΗ ΜΕΤΑΛΛΙΚΩΝ ΠΛΑΙΣΙΩΝ ΜΕ ΛΕΠΤΟΜΕΡΗ ΠΡΟΣΟΜΟΙΩΜΑΤΑ ΠΕΠΕΡΑΣΜΕΝΩΝ ΣΤΟΙΧΕΙΩΝ

Αποστόλης Α. Βράκας, M.Sc.
Αριστείδης Γ. Παπαχρηστίδης, Ph.D.
Μανόλης Παπαδρακάκης, Καθηγητής
Εθνικό Μετσόβιο Πολυτεχνείο
Πολυτεχνειούπολη Ζωγράφου, Αθήνα 15780, Ελλάδα
apostolis.vrakas@gmail.com

ΠΕΡΙΛΗΨΗ

Σκοπός αυτής της εργασίας είναι η μη γραμμική ανάλυση με λεπτομερή προσομοιώματα πεπερασμένων στοιχείων μεταθετών μεταλλικών πλαισίων αποτελούμενων από κοχλιωτούς κόμβους δοκού-υποστυλώματος με προεξέχουσα μετωπική πλάκα. Αρχικά, πραγματοποιείται προσομοίωση των κόμβων: α) με επιφανειακά στοιχεία κελύφους σε συνδυασμό με ραβδωτά στοιχεία δοκού, και β) με εξαεδρικά-οκτακομβικά τρισδιάστατα πεπερασμένα στοιχεία. Λαμβάνονται υπόψη τόσο μη γραμμικότητες υλικού όσο και γεωμετρικές μη γραμμικότητες συμπεριλαμβανομένων των επαφών μεταξύ των διαφόρων συστατικών μερών των συνδέσεων. Υπολογίζονται καμπύλες ροπής-στροφής (M-φ) χαρακτηριστικών κόμβων υποβαλλόμενων σε στατικά φορτία, και συγκρίνονται με πειραματικά αποτελέσματα και προβλέψεις του EC3 για την αποτίμηση των διαφόρων αριθμητικών μοντέλων. Ακολουθεί η ανάλυση πολυώροφων μεταλλικών πλαισίων με χρήση υβριδικών αριθμητικών μοντέλων. Για τα δομικά μέλη χρησιμοποιούνται ραβδωτά στοιχεία δοκού-υποστυλώματος, ενώ οι κόμβοι προσομοιώνονται λεπτομερώς με θεώρηση κατάλληλων κινηματικών εξαρτήσεων στις διεπιφάνειες. Πραγματοποιούνται μη γραμμικές στατικές αναλύσεις υπό οριζόντια φορτία (pushover) επιδεικνύοντας την επιρροή των κόμβων στη συμπεριφορά του φορέα. Τέλος, εξετάζεται η επίδραση της διάτμησης στον κορμό του υποστυλώματος μέσω ενίσχυσης του με πρόσθετα ελάσματα.