WARSAW UNIVERSITY OF TECHNOLOGY Index 351733
FACULTY OF CIVIL ENGINEERING
COMMITTEE FOR CIVIL AND WATER ENGINEERING
POLISH ACADEMY OF SCIENCES ISSN 1230-2945

DOI: 10.24425/ace.2022.143039

ARCHIVES OF CIVIL ENGINEERING

Vol. LXVIII ISSUE 4 2022



**pp. 293 – 309**



**Research paper**

# Analysis of steel frame under selected accidental situation

**Damian Kukla[1], Aleksander Kozlowski[2]**

**Abstract:** The current design codes recommend designing the building structures based on the assumption avoiding the disproportionate to the initial cause damage during accidental situation. As a main strategy to mitigate a progressive collapse an alternative load path method is recommended. Flush and extended bolted end-plate joint to connect beam-to-column were experimentally tested. Hierarchical validation of joint FEM models based on experimental test results were performed. The numerical dynamic analysis by finite element method of selected steel frame under column loss scenario is presented. The planar 2D model of frame were used. Shell elements for beams and columns and solid elements for joints were employed respectively. Nonlinear material and geometry were applied in the analysis. Johnson-Cook model was used to describe the change of steel parameters by dynamic Increase Factor (DIF). The Rayleigh model to include the damping effects in the analysis was used. The dynamic analysis was performed with the use of Abaqus/Explicit module. Main conclusion of presented research it that to achieve the required level of robustness, bolted beam-to-column joints with extended end-plate of thickness more than 15 mm should be used.

**Keywords:** FEA, dynamic analysis, steel end-plate bolted joints, accidental situation, column loss, robustness

[1]PhD. Eng. Rzeszów University of Technology, Faculty of Civil and Environmental Engineering and Architecture, Poznańska 2, 35-084 Rzeszów, Poland, e-mail: damian9105@o2.pl, ORCID: 0000-0003-1612-8394

[2]Prof. DSc., PhD, Eng., Rzeszów University of Technology, Faculty of Civil and Environmental Engineering and Architecture, Poznańska 2, 35-084 Rzeszów, Poland, e-mail: kozlowsk@prz.edu.pl, ORCID: 0000-0003-0920-6681

# 1. Introduction

The subject of numerical evaluation of the resistance of steel framed structures is intensively developed issue. The researchers' and designers' approach in this area can be divided into two basic ones. The first is performed by static analysis, which is dominate in every day practice. The second one is dynamic analysis of structures, applied in earthquake and robustness analysis. Several items in this field can be found in the scientific literature.

The numerical dynamic study on progressive collapse of planar steel frames under sudden column loss were presented in [18]. Finite element models with refined shell elements were developed. The welded beam-to-column joints were applied. It was found that frames with strong beams and weak columns have a higher robustness of collapse than frames with strong columns and weak beams.

In [24], a multi-spring model (MSC) was created as a joint model to assess the rotational stiffness of a semi-rigid connection to be used in the analysis in column losses scenario. The effects of a semi-rigid connection were observed as crucial for the deformation and load distribution. The greatest maximum displacement was observed at the top of the removed column for the loss of the outer column.

The influence of brick filling walls on the mechanism of the progressive catastrophe caused by the removal of the central column of the steel frames is presented in [20]. The analyzes were performed using the finite element method. A significant increase in the initial stiffness and the maximum applied load was achieved. On the other hand, there has also been a reduction in the ductility of the steel frames.

The analysis of robustness steel moment frame under vehicle impact were presented in [10]. A various speed of vehicle was adopted to assessing structural response. The collision to the corner column caused much greater consequences than collision to the exterior column.

The study of the progressive collapse of flat steel frames under the effect of column losses was presented in [11]. The influence of the modeling technique and finite element analysis was analyzed. The mesh size and the removal time of the column were indicated as having the major influence on the structural response.

In [6] global behavior of flat steel frames using traditional beam-to-column joints and innovative joints in the event of loss of a central column was analyzed. Extended end-plate joints were used as traditional joints, while the FREEDAM joints tested in [2] were introduced as innovative. As the main results significant benefits in terms of joints deformability due to use of innovation joints was achieved.

The analysis of the robustness of a five-story planar steel frame structure during sudden removal of a column was presented in [21]. Several locations for the removal of the column from the structure was assumed. The greatest collapse consequences was observed when the corner column was removed.

The behaviour of the steel frame designed to seismic impacts with joints specially designed for this type of structure was analyzed numerically in [22], in terms of the column removal scenario. The excellent strength and high level of safety of the frames with these connections have been noted.

Numerical tests of the resistance of a multi-storey steel structure building in the event of loss of one or more columns are presented in [3]. The type of beam-to-column connection and the interaction between the concrete slab and the steel beams were analyzed as main factors. Static and dynamic nonlinear analysis were used to test the structure response. High resistance to exceptional situations was found in structures designed for seismic impacts.

In [4] an analysis of resistance and sensitivity to the progressive collapse of a typical steel frame building, subject to a scenario of an accidental explosion or a terrorist attack was presented. The detonation of a 500 kg load when it is close to one of the outer columns may cause the building to collapse.

A simplified method of assessing collapse resistance of multi-storey buildings exposed to sudden loss of a vertical load-bearing element was presented in article [23]. Two possible scenarios were investigated, i.e. removal of an outer or corner column in a typical steel-concrete composite building. Buildings with typical structural configurations were vulnerable to progressive collapse initiated by the loss of a column. This susceptibility is mainly related to the size of the beam spans needed to safely transfer the instantaneous gravity loads to the remaining undamaged structure, as well as to the types of connection used at the beams ends.

The numerical analysis of beam-to-column joint behavior in different design situations were presented in [7]. The validation of numerical analysis results with experimental results for joints under hogging and sagging moments were shown. The complex methodology to determine joints performance was used.

A study on numerical modeling of steel and steel-concrete composite typical joints in framed structures for the estimation of the global robustness of multistory buildings were presented in [17]. Nonlinear dynamic computer simulations in software LS-DYNA were conducted. The developed plan for hierarchical verification and validation in numerical analysis was used.

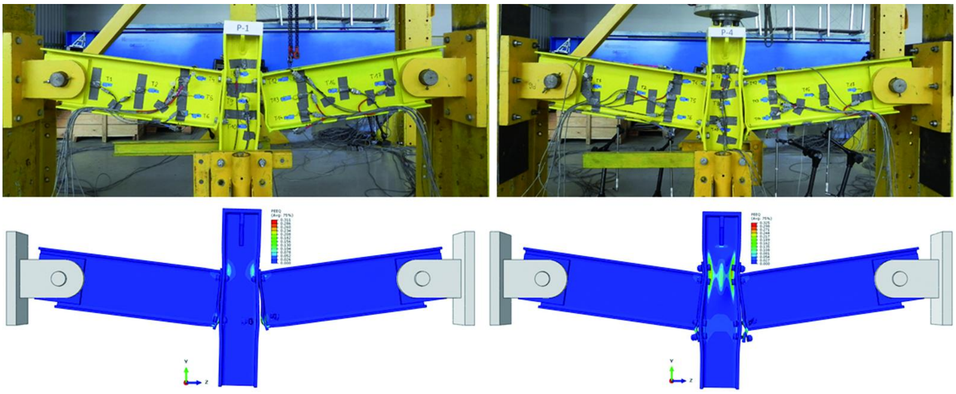
The behavior of steel and steel-concrete composite joints as well as proposition of including the postlimiting behavior in the joints characteristic was presented [8]. The accuracy of the analytical model was compared with the results of experimental studies. To verify the analytical model, experimental tests on three isolated samples of beam-to-column connections under hogging bending moment were used.

## 2. Analysis of joints

The aim of presented research is to analyze the behavior of steel framed structures in scenario of column removal. Analysis was divided on the following steps:

- experimental test of the most often used in skeleton structures joints, to observe its behavior under large displacements,
- advanced FEM modelling of experimentally tested joints, their hierarchical validation,
- analysis and validation of FEM joint models of subframe tested experimentally,
- dynamic analysis of steel frame, using validated joints models.

Experimental tests on steel joints were carried out as the first step in the robustness analysis of framed structures. The six bolted end-plate joints with flush and extended end-plate were tested. The details of experimental test of double side end-plate joints were presented in [13]. On the basis of these tests results, the numerical analysis and validation of finite element models of these joints were done, which were presented in [16]. Thanks to the use of hierarchical validation, strict compliance of both techniques was obtained at the level of results and graphical comparison (Fig. 1). Experimental tests and numerical analysis allowed to obtain a lot of information about the behavior of the steel end-plate joints in accidental situations as a column loss scenario.



(a) (b)

Fig. 1. Comparison views of experimental test [13] and numerical analysis [16] of joint: a) 10 mm flush end-plate, b) 10 mm extended end-plate

The performed models of end-plate joints were used to conduct the parametric analysis of different factor influencing the rotational and ultimate capacity of the joints. The results of these analysis were presented in [15].

## 3. FEM analysis of frame substructure

The next step was the numerical analysis [14] of the frame subsystem (Fig. 2), based at experimental tests made as part of the research work [12, 19]. The analyzes were performed on two models. The first one concerned a substructure with flush end-plate joints with a 10 mm plate thickness. The second case involved the same subsystem but with extended 10 mm end-plate joints. Both systems were modeled in the same way.

For this purpose, the analysis of this systems (Fig. 2) were conducted in the Abaqus software [1]. Explicit dynamic integration method to overcome convergence difficulties in static solver were used. To produce quasi-static response a time step control was applied. Mass scaling as a factor to reduce computational time of analysis was also used. The frame substructure (Fig. 3a) was modeled using two types of finite elements. Parts of

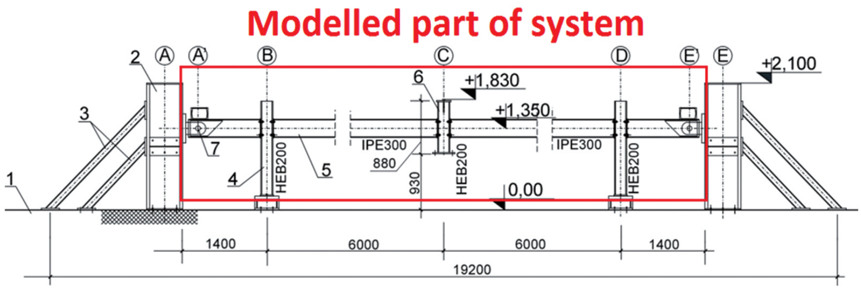
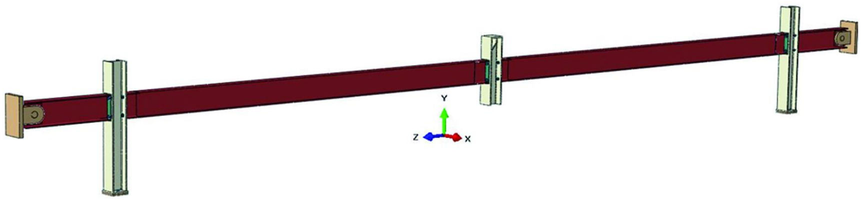



Fig. 2. Planar view of frame substructure tested in [12, 19]



(a)

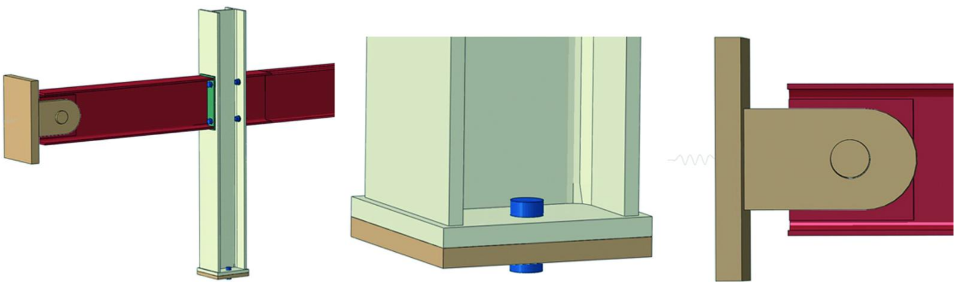
(b)

Fig. 3. 3D view of substructure with flush end-plate joint: a) model [14], b) specimen under test [12,19]

subframe, with the highest values of strains, as end-plates, end parts of beams, columns, bolt connectors and pin connections at end of system, were modeled by solid elements C3D8R.

The parts of beams between external and internal column (of about 5 m length) were modeled by shell elements S4R. Application of shell elements at long beam elements allows to significant reduction of element numbers and of computational time of the analysis. Both elements allows to large displacements and strains at analysis time. To repro-duce contact between elements a general contact interaction was predicted. A hard contact to simulate the unilateral contact in normal direction of the interface between connected elements was used. An isotropic friction coefficient at value equal 0.3 to represent tangential behavior was applied.

Appropriate boundary conditions representing those in the experimental studies (Fig. 3b) were applied. The upper and lower parts of the central column (not vertically supported) were laterally supported (blocked movement in the model plane) in order to prevent inclination out of the model plane, as was done during experimental tests. The connection of the model to the test setup in a simplified way was modeled. The external connections of test setup (Fig. 4a) in the form of plates with pins were modeled. Horizontal movement parallel to the long side was allowed but limited by the horizontal stiffness of the supports. The horizontal spring of supports was modeled (Fig. 4c) with a constant linear behavior and the value of 450 kN/mm. In addition, the outer columns of the model were laterally supported (perpendicular to the model plane). An additional base plate (Fig. 4b) marked in brown was modeled as elements of the test laboratory floor. In order to connect the column base plate (Fig. 4b), two M20 bolts were used.



(a) (b) (c)

Fig. 4. Detailed view of: a) external column, b) column base, c) external pinned connection

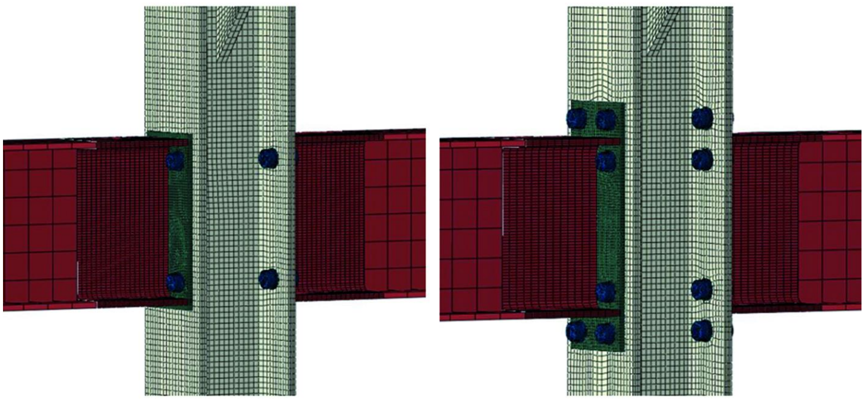
Three steps of loading procedure were used in the analysis. In the first step the axial compression force 10 kN was applied to bolts to simulate initial contact between column flanges and end-plates. The gravity load of whole model as second step were applied to reduce dynamic effects and achieving model stability. In third step the vertical load on top of internal column was applied by carrying out a displacement controlled dynamic pushdown analysis. Smooth step option was applied to reduce dynamic effects.

As a columns cross-section HEB 200 was assumed while IPE 300 was used to modeling beams. The flush and extended end-plate with 10 mm thickness were applied as two types of end-plate. The reinforcing ribs and horizontal plate at top of central column was used at place of load application. The welds at whole model were neglected. A M20 bolts were used to connect the elements of the joints. A bolt, washer and nut were modeled as parts of connector. The details of modeling bolt connector was presented in [16].

Corresponding material models were used for each of the model elements. The material properties of HEB 200 and IPE 300 were taken from [12, 19]. Due to lack of material tests of end-plate and bolts, values obtained from own tests [13] were assumed. The true stress-strain relationship was defined to simulate realistic behavior on material models.

Elasto-plastic material model with ductile damage was assumed for analysis. When specific equivalent plastic strain at the onset is reached, the model assumes to start of damage. A damage evolution was defined to progressive degradation of material stiffness. Damaged elements will be deleted from the mesh, when the fracture strain is reached. Additionally, a displacement type with linear degradation for damage evolution was applied. For the pinned connection at both ends of the model (i.e. pins and plates), the elastic model was adopted as for steel S235.

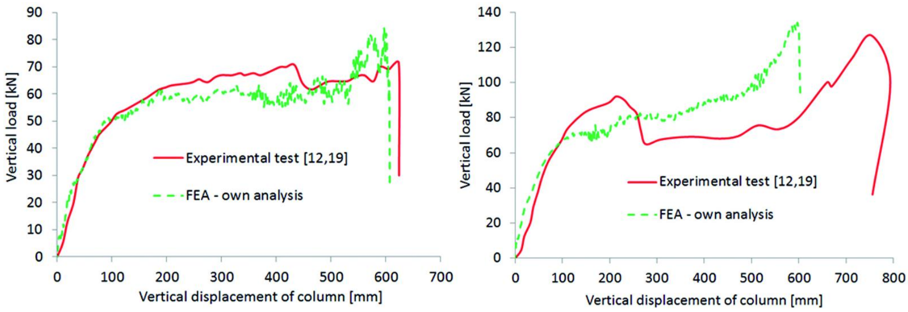
For solid elements the hexagonal element shape in the mesh control were used. To obtain regular shapes of mesh in all solid elements of model the sweep technique with medial axis algorithm were developed. The columns, ends of beams and end-plate were meshed by 10 mm and 5 mm mesh size (Fig. 5), respectively. The bolt connector was meshed singly. As a main density equal 3.5 mm for bolts and nut were assumed. To generate quadrilateral elements of mesh a quad-dominated element shape option with free technique in medial axis algorithm was used for shell elements. As a size (Fig. 5) of mesh 50 mm density was assumed.



(a) (b)

Fig. 5. Detailed view of: a) flush, b) extended end-plate joint with mesh

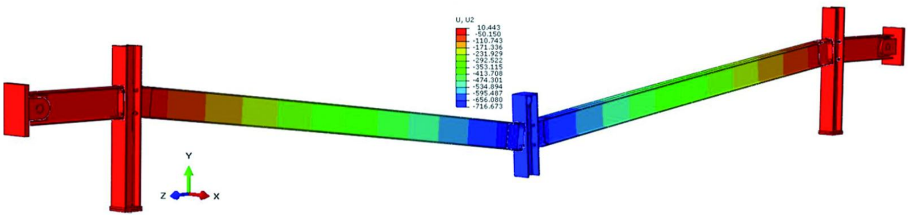
In order to validate the numerical analysis results based on the finite element method, the relevant results were compared with experimental tests of the frame subsystem with bolted joints, with a 10 mm thick flush and extended end-plate with. The vertical load – vertical displacement curve of the subsystem was assessed for compliance of both methods. The images of deformation and joint failure models were graphically compared. The force-displacement curve obtained from the experimental and numerical tests of frame with flush end-plate joints are shown in Fig. 6a. It can be noticed that up to a specimen deflection of about 200 mm the compatibility is almost perfect. After exceeding the displacement of 200 mm, there is a slight increase in the difference of both results. To damage of model a good agreement of final displacement in both cases were achieved. The response curves of the steel subsystem with extended end-plate joints are shown in Fig. 6b. A very good convergence was obtained until the vertical displacement of the column was about 100 mm.



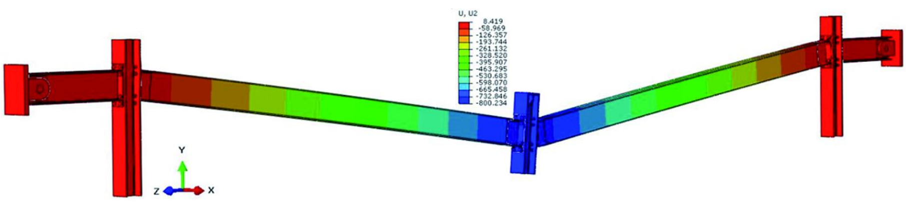

(a) (b)

Fig. 6. Response curve load-vertical displacement of specimen with: a) flush end-plate, b) extended end-plate

After exceeding the deflection of 100 mm, the values began to differ slightly. The difference in the final displacement on failure was about 150 mm. In Figure 7 the maps of vertical displacements of the entire analyzed subsystem are presented. The highest deflection values in the middle of the model is also shown. The largest deformations of the system were located at the joints.



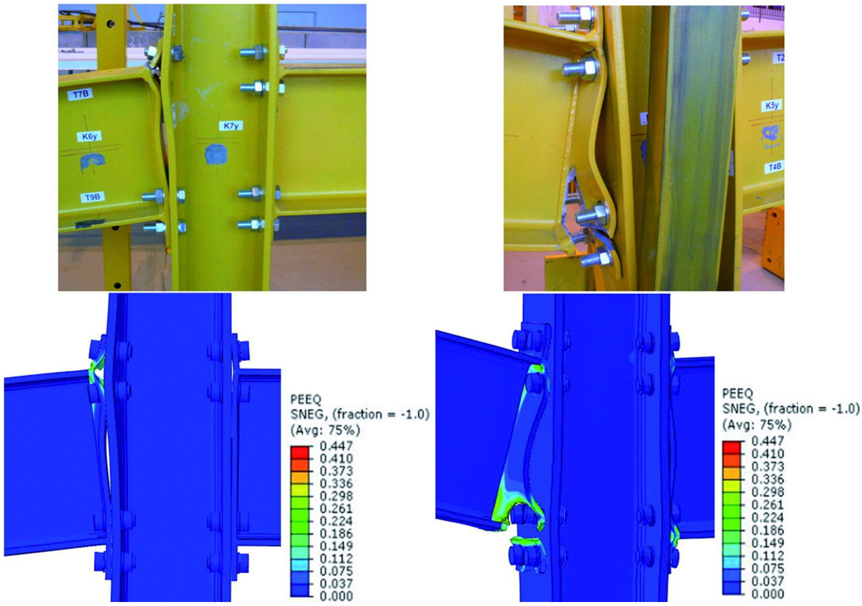
(a)



(b)

Fig. 7. Map of vertical displacement of model with: a) flush end-plate, b) extended end-plate

The inner joint of the outermost column was significantly deformed. Large deformations (Fig. 8a) especially in the end-plate were reached. The bend of the column flange is also visible. The good agreement at comparative image of the deformation of external column is shown.




(a) (b)

Fig. 8. Failure mode of experimental test [12, 19] and numerical analysis [14] of steel subframe with: a) external column joint, b) internal column joint

The deformation of the middle column joints is presented in the Fig. 8b. The significant deformation of the joint (Fig. 8b) was also obtained in the numerical analysis. The good agreement of comparison (Fig. 8b) of the joint views after the tests was confirmed. Both joints were destroyed by fracture the end plate at the level of the beam bottom flange.

# 4. Dynamic analysis of the frame

Based on the results of previous tests and numerical analyzes performed on plane systems, a dynamic analysis of a flat steel frame was carried out.

The sections of IPE 300 and HEB 200 were used as cross-sections of beams and columns respectively. In order to assess the resistance of the frame to collapse, experimentally tested joints were adopted, i.e. joints with extended end-plate in configuration of plate thickness equal to 15 mm.

The diagram of the main transverse system and its dimensions is presented in Fig. 9. The column spacing in the axis of 5.0 m and the story height in the axis of the beams to 3.5 m were assumed. The height of the structure was limited to 3 storeys due to the use of the same column cross-section at the height of the building and avoiding the use of an additional column splices. Spacing of the main transverse systems at 5.0 m was assumed.

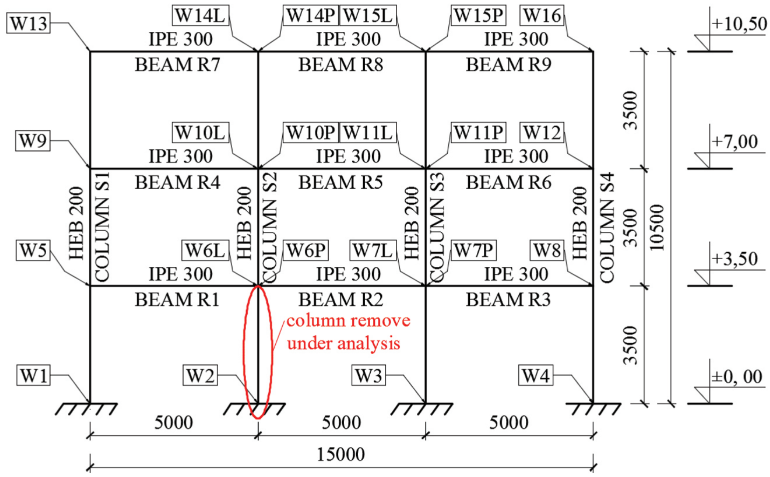



Fig. 9. Static diagram of the analyzed frame

It was assumed that the columns in the base (nodes W1, W2, W3, W4) were fully restrained. The base restraint was released in due course of the analysis as the column removal scenario, to simulate an accidental situation. The beam-to-column joints were assumed to be the same throughout the entire system. On the upper surface of each beam horizontal displacement blocks were installed in the direction perpendicular to the main transverse systems, which was to simulate the presence of the slab of each storey. This blockade protects the beam from lateral-torsion buckling during the analysis. Reduction numbers of finite elements used to model framed system due to large size of numerical model was done. For this purpose two types of finite elements were used. Shell elements S4R were used for sections of beams between joints over a length of about 4 m and sections of columns between joints over a distance of about 2.5 m. The joints, as elements with the greatest stresses and strains, were modeled using C3D8R solid element type in the same way as described in the previus chapter. In this way significantly reduce the computational time of analysis was obtained. The use of detailed models of joints built of solid elements let to obtain accurate analysis results, as actual image of joits behaviour and failure models.

Due to the dynamic analysis with the sudden removal of the internal column Dynamic Increase Factor (DIF) for the mechanical parameters for the bolts and end-plates was applied. For this purpose Johnson-Cook model was used to describe the change of steel parameters. Remaining material models used in numerical analysis as obtained from experimental test of joints were adopted.

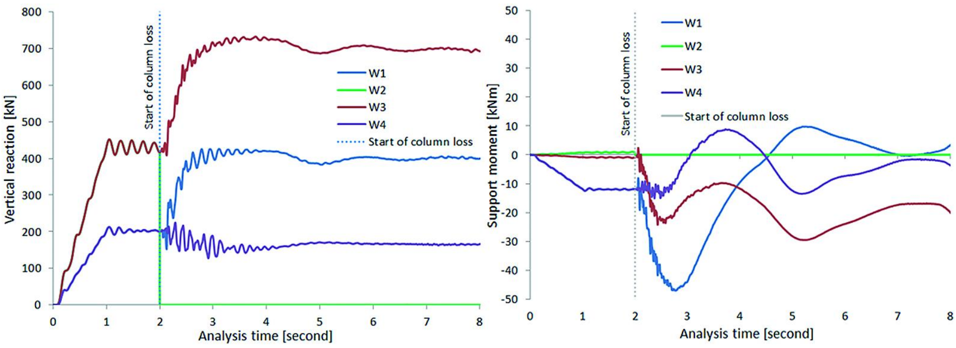
The Rayleigh damping model was used to include the damping effects in the analysis. The Abaqus/Explicit module was used to simulate the dynamic analysis of column loss in numerical simulation. In dynamic analysis the loading as mass were applied. The corresponding material density equal to 7850 kg/m$^3$ for steel elements were assumed. Due to fact that beams as elements that take over the dead load and live load from the floor slab the beam material density was scaled appropriately – density of beam material takes into account a mass of the slab and the beam's own weight. Damping at the level of 5% of mass in the case of dynamic analysis of robustness as recommended in GSA [9] was assumed.

Appropriate loading procedure were used to generate the response of the frame systems. The loading process was divided into 5 steps:

– initial, which was a default step introduced by the program to implement the applied boundary conditions,
– bolt pressure to simulate the initial tightening of the bolts, which consisted of applying a double-sided load of 10 kN to all bolts of each joint in order to obtain the initial contact of the joint surface,
– specific load in the form of applied gravity load control with the option of a constant linear incremental due to reduce dynamic effects to a minimum in the loading phase in inital analysis. At the time between 1$^{\text{st}}$ to 2$^{\text{nd}}$ second the stabilization phase was assumed with full loading stage. Thereafter, the load was kept constant until the end of the analysis. It was assumed that the dead load on the floor surface resulting from the arrangement of structural elements and finishing layers was 4.00 kN/m$^2$. The live load per floor area, resulting from the function of a residential building, was assumed to be 2.00 kN/m$^2$ and the load from partition walls to 0.8 kN/m$^2$. In total, the live load was assumed to 2.8 kN/m$^2$. Load combinations under exceptional situation according to standard [5] were applied.
– loss of a column was simulated, as the loss of restraint at the "0" level for central column node W2 (Fig. 9).

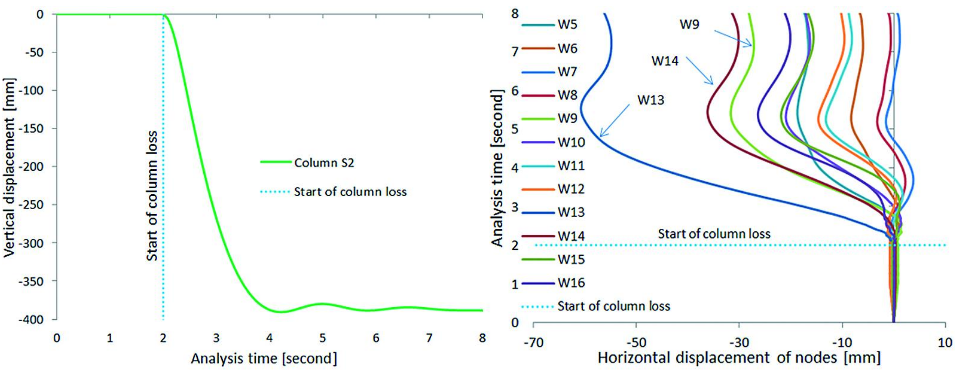
In this section the selected results of frame analysis are presented. In Figure 10 the curves of the support moments and vertical support reactions of three bays frame with extended end-plate joints under sudden loss of column are presented. The development of the vertical supports reaction at the nodes W1 to W4 are shown in Fig. 10a. Until the middle column was removed, the vertical reactions of all support joints increased. Sudden dynamic response of the structure resulted by the remove of the support of the W2 joint up to about 3rd second was observed. In the time from 3rd second to 7th second, the vertical support reactions stabilized. It is worth noting that the vertical reaction of the node W1 and W3 after the removal of the column increased its value, while the value of the reaction in node W4 decreased. The support moments of nodes W1 to W4 (Fig. 10b) changed significantly during the analysis. At the initial load period, a negative value of the support moments were observed. After the support of the W2 node was removed, the moment value changed, resulting from the dynamic response of the structure. Next, the dynamic effects stabilized, while the sign of the moment at W1 node changed at $4^{th}$ second. About $7^{th}$ second the stabilization of the support moments to the end of analysis were observed.

The vertical displacement of the S2 column during the analysis is presented in Fig. 11a. After the column was removed, the displacement value increased significantly. In about $4^{th}$ second, the maximum value of the vertical translation was reached, whereafter the increment was stopped until the end of the analysis. Horizontal displacements of the nodes during the analysis are presented in Fig. 11b. Until $2^{nd}$ second, the displacement values were negligible. The removal of the S2 column initiated the development of horizontal displacements of the system, especially on the left side of the removed column. The greatest values of displacements were recorded at the nodes W13, W14 and W9.




(a) (b)

Fig. 10. Response curves of three bays frame with 15 mm thickness extended end-plate joint under sudden column loss: a) vertical reaction – time analysis, b) support moment vs. time analysis



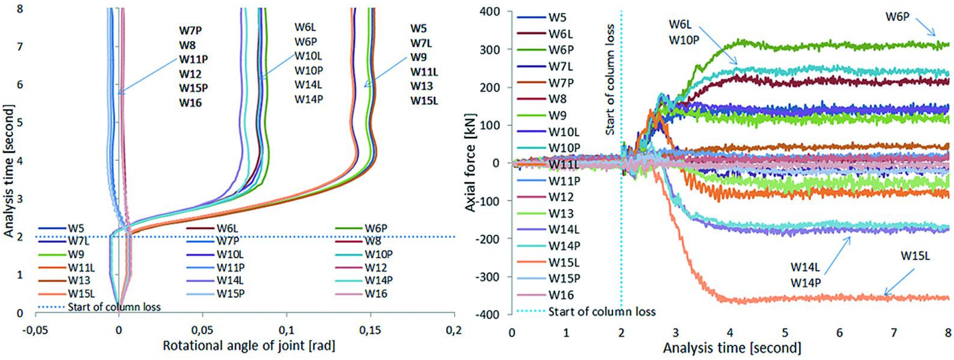

(a) (b)

Fig. 11. Curves of: a) vertical displacement – time analysis, b) time analysis vs. horizontal displacement of three bays frame

The joints behavior during the analysis based on the time-rotation curves of the joints are shown in Fig. 12a. In the time until the column was removed, the rotations of the joints were negligible. After the removal of the column, development of rotations were observed. After 4$^{\text{th}}$ second, a clear division of joints into three groups were noticed.

To the first group the joints with the smallest rotation values were ranked, i.e. joints on the right side of the S3 column. To second group were classified joints with average values of rotations, which include the joints adjacent to the S2 column. Into the last group of joints with the greatest rotations, including the joints of the S1 column and on the left side of the S3 column were included.

Until the beginning of the column loss (Fig. 12b), the values of the axial force in the joints were negligible. After exceeding the time of the $2^{nd}$ second, the development of axial forces were observed. The differences in the values of axial forces in the structure

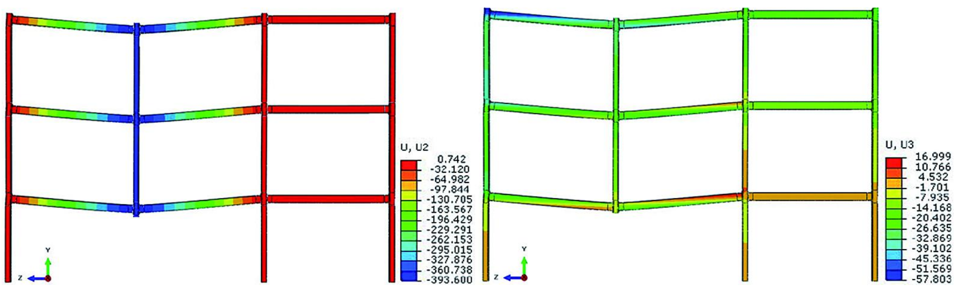



(a) (b)

Fig. 12. Curves of: a) time analysis – rotational angle of joints, b) axial force – time analysis

were significant, ranging from significant tensile forces to significant compressive forces. The highest value of the axial tensile force was recorded in the joint W6P, W6L and W10P, while the highest value of the compressive force in the joint W15L.

In Figure 13 the global displacement maps of the frame structure under sudden column loss are shown. The maximum vertical displacement (Fig. 13a) at column S2 was reached. In case of horizontal displacement (Fig. 13b), the highest value at top of S1 column was obtained.




(a) (b)

Fig. 13. View of frame under sudden column loss at final stage of analysis: a) vertical displacement, b) horizontal displacement

In Fig. 14 are presented the maps of strains and deformations and the views of joints of the first story under end time of the analysis. A significant deformation of external joint W5 and internal joints W6 and W7 can be observed. The considerable strains of column web at the height between the upper and lower beam flanges were obtained in W5 joint. The maximum strains in the W6 joint was concentrated in level of bottom beam flange.

In joints of the +10.50 level (Fig. 15) significant deformations were also observed, especially of the column flanges and the end-plate, similar to the +3.50 level. Comparing

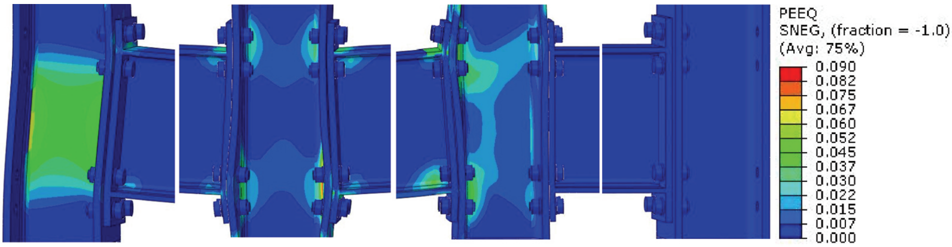



Fig. 14. Map of strains of joint (from left W5 to W8) with 15 mm thickness exteneded end-plate at level of structure +3.50 at final stage of analysis

the corresponding joints of the S3 column (i.e. joint W7 in Fig. 14 and W15 in Fig. 15), a clearly different pattern of column web deformations at the height of the joints was noticed.

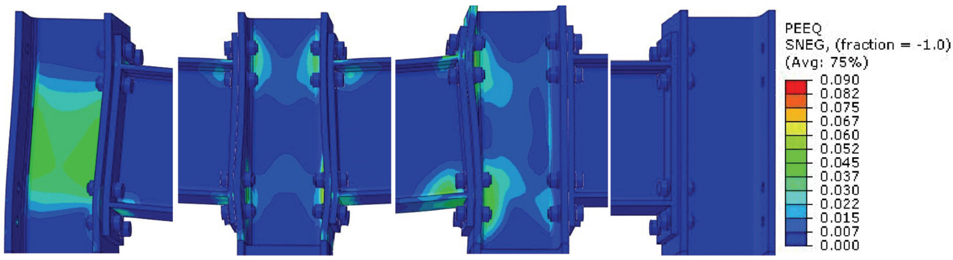



Fig. 15. Map of strains of joint (from left W13 to W16) with 15 mm thickness exteneded end-plate at level of structure +10.50 at $8^{th}$ second of analysis

The required level of robustness to a catastrophe in the event of a sudden loss of the internal column in case of analysed frame was obtained. The redistribution of forces in the structure and the creation of secondary mechanisms at a level sufficient to stop the collapse was reached by the use in structure end-plate joints with extended end-plate of 15 mm thickness.

## 5. Conclusions

The numerical investigation of steel frame with end-plate joints in column loss scenario was conducted. The full advanced model with solid and shell elements was created. The planar frame diagram with three storey and three bays was analyzed. Extended end-plate joints as a connection of beam-to-column were adopted to whole model. Based on the analysis, the following conclusions can be presented:

– in order to achieve the required level of robustness, the joints of the structure must have adequate rotation capacity and the ability to transfer also tensile axial forces,

– the use of bolted extended end-plate joints leads to creation of a mechanism preventing a progressive collapse,
– in the case of frames with exteneded end-plate joints, the distribution of axial forces at individual levels of the structure is uneven, taking into account the appearance of tensile and compressive axial forces,
– an important aspect from the point of view of the robustness of the structure is the phenomenon of catenary action, in which the values of the tensile axial force reach about 350 kN,
– to achieve the required level of robustness to a progressive collapse, bolted beam-to-column joints with extended end-plate of thickness more than 15 mm should be used,
– the joints in the close vicinity of the removed column have a significant impact to resist collapse of frame structures.

## References

[1] Dassault, *Abaqus – User Analysis User's Manual*, *Dassault Systems Simulia Corp*.

[2] M. D'Antimo, M. Latour, G. Rizzano, J-F. Demonceau, J-P. Jaspart, "Preliminary study on beam-to-column joints under impact loading", *The Open Construction and Building Technology Journal*, 2018, vol. 12, pp. 112–123; DOI: 10.2174/1874836801812010112.

[3] F. Dinu, D. Dubina, I. Marginean, "Improving the structural robustness of multi-story steel-frame buildings", *Structure and Infrastructure Engineering*, 2015, vol. 11, no. 8, pp. 1028–1041; DOI: 10.1080/15732479.2014.927509.

[4] H.M. Elsanadedy, T.H. Almusallam, Y.R. Alharbi, Y.A. Al.-Salloum, H. Abbas, "Progressive collapse potential of a typical steel building due to blast attacks", *Journal of Constructional Steel Research*, 2014, vol. 101, pp. 143–157; DOI: 10.1016/j.jcsr.2014.05.005.

[5] *EN 1990 Eurocode – Basis of structural design*. European Committee for Standardization CEN Brussels, 2002.

[6] A.B. Francavilla, M. Latour, G. Rizzano, J-F. Jaspart, J-F. Demonceau, "On the Robustness of Earthquake-Resistant Moment-Resistant Frames: Influence of Innovative Beam-to-Column Joints", *The Open Construction and Building Technology Journal*, 2018, vol. 12, pp. 101–111; DOI: 10.2174/1874836801812010101.

[7] M. Gizejowski, L. Kwasniewski, B. Saleh, M. Balcerzak, "Numerical study of joint behaviour for robustness assessment", *Applied Mechanics and Materials*, 2012, vol. 166-169, pp. 3114–3117; DOI: 10.4028/www.scientific.net/AMM.166-169.3114.

[8] M.A. Gizejowski, B.A.A.K. Saleh, W. Barcewicz, J. Uziak, "Effect of postlimiting behaviour of isolated beam-to-column joints observed in force controlled experiments", *Journal of Civil Engineering, Environment and Architecture*, 2013, vol. 30, no. 60(2), pp. 27–50.

[9] *General Services Administration. Alternate Path Analysis & Design Guidelines for Progressive Collapse Resistance*. October 24, 2013.

[10] H. Kang, J. Kim, "Progressive collapse of steel moment frames subjected to vehicle impact", *Journal of Performance of Constructed Facilities*, 2014 vol. 29, no. 6; DOI: 10.1061/(ASCE)CF.1943-5509.0000665.

[11] F. Klakojour, M.R. Sheidaii, "Effects of finite modelling and analysis techniques on response of steel moment-resisting frame in dynamic column removal scenarios", *Asian Journal of civil Engineering*, 2018, vol. 19, pp. 295–307; DOI: 10.1007/s42107-018-0027-2.

[12] A. Kozlowski, M. Gizejowski, L. Ślęczka, et al., "Experimental investigation of the joints behaviour–Robustness assessment of steel and steel concrete composite frames", in *Proceeding of the 6th European Conference on Steel and Composite Structures*. Budapest, 2011, pp. 339–344.

[13] A. Kozlowski, D. Kukla, "Experimental tests of steel unstiffened double side joints with flush and extended end plate", *Archives of Civil Engineering*, 2019, vol. 65, no. 4, pp. 127–154; DOI: 10.2478/ace-2019-0051.

[14] D. Kukla, A. Kozlowski, "Analysis of steel bolted end-plate joints under accidental situation", in *Modern Trends in Research on Steel, Aluminium and Composite Structures*, M. Gizejowski, et al., Eds. Routledge, 2021; DOI: 10.1201/9781003132134-40.

[15] D. Kukla, A. Kozlowski, "Parametric study of steel flush and extended end-plate joints under column loss scenario", *Engineering Structures*, 2021, vol. 237, art. ID 112204; DOI: 10.1016/j.engstruct.2021.112204.

[16] D. Kukla, A. Kozlowski, T. Siwowski, "Numerical analysis of steel double side joints with flush and extended end plate under accidental situation", *Ce/papers,* 2021, vol. 4, no. 2-4, pp. 77–84; DOI: 10.1002/cepa.1265.

[17] L. Kwasniewski, M. Gizejowski, "Numerical modelling of joint ductility in steel and steel-concrete composite frames", in *SDSS'Rio 2010 STABILITY AND DUCTILITY OF STEEL STRUCTURES, Rio de Janeiro, Brazil, 8-10 Sept. 2010*, E. Batista, P. Vellasco, L. de Lima, Eds. Rio de Janeiro, 2010, pp. 239–246.

[18] L-L. Li, G-Q. Li, B. Jiang, Y. Lu, "Analysis of robustness of steel frames against progressive collapse", *Journal of Constructional Steel Research*, 2018, vol. 143, pp. 264–278; DOI: 10.1016/j.jcsr.2018.01.010.

[19] B.A.A.K. Saleh, "Modelling of beam-to-column joints of Steel-Concrete Composite Frames Subjected to Standard and Extreme Load Combinations", PhD thesis, Faculty of Civil Engineering, Warsaw University of Technology, 2013.

[20] S. Shan, S. Li, S. Wang, "Effect of infill walls on mechanism of steel frames against progressive collapse", *Journal of Constructional Steel Research*, 2019, vol. 162, art. ID 105720; DOI: 10.1016/j.jcsr.2019.105720.

[21] H.R. Tavakoli, F. Kiakojouri, "Numerical Study of Progressive Collapse in Framed Structures: A New Approach for Dynamic Column Removal", *International Journal of Engineering*, 2013, vol. 26, no. 7, pp. 685–692.

[22] G. Vasdravellis, M. Baiguera, D. Al.-Sammaraie, "Robustness assessment of a steel self-centering moment-resisting frame under column loss", *Journal of Constructional Steel Research*, 2018, vol. 141, pp. 36–49; DOI: 10.1016/j.jcsr.2017.11.004.

[23] A.G. Vlassis, B.A. Izzuddin, A.Y. Elghazouli, D.A. Nethercot, "Progressive collapse of multi-storey buildings due to sudden column loss – Part II: Application", *Engineering Structures*, 2008, vol. 30, no. 5, pp. 1424–1438; DOI: 10.1016/j.engstruct.2007.08.011.

[24] J. Znang, J. Jiang, S. Xu, Z. Wang, "An investigation of the effect of semi-rigid connections on sudden column removal in steel frames", *Structures*, 2018, vol. 13, pp. 166–177; DOI: 10.1016/j.istruc.2017.12.001.

# Analiza stalowej konstrukcji ramowej w wybranej sytuacji wyjątkowej

**Słowa kluczowe:** metoda elementów skończonych, połączenia śrubowe, stalowe węzły doczołowe, sytuacja wyjątkowa, usunięcie słupa

## Streszczenie:

Obecne przepisy projektowe zalecają projektowanie i wykonywanie konstrukcji budowlanych w oparciu o unikanie nieproporcjonalnego do pierwotnej przyczyny uszkodzenia w sytuacji wyjątkowej. W obecnych normach projektowych przedstawiono kilka sposobów ochrony konstrukcji przed katastrofą postępującą. Jako główną strategię łagodzenia katastrofy postępującej zaleca się metodę alternatywnej ścieżki obciążenia. Odporność konstrukcji jest definiowana jako zdolność konstrukcji do zatrzymania katastrofy. Wykonano badania doświadczalnie śrubowych węzłów doczołowych belka-słup z blachą wpuszczoną i wystającą. Przeprowadzono walidację hierarchiczną modeli MES węzłów na podstawie wyników badań eksperymentalnych. Przedstawiono dynamiczną analizę numeryczną, opartą na metodzie elementów skończonych, wybranej ramy stalowej w sytuacji wyjątkowej. Zastosowano płaski model konstrukcji. Do budowy modelu ramy wykorzystano odpowiednio elementy powłokowe dla belek i słupów oraz elementy bryłowe dla węzłów. Wykorzystano nieliniowy

model materiału i geometrii do wykonania modelu. Charakterystyki materiałowe stali zostały zmodyfikowane przez zastosowanie dynamicznego współczynnika wzrostu (DIF), przy użyciu modelu Johnson-Cook. W obliczeniach wykorzystano moduł Abaqus/Explicit z uwzględnieniem tłumienia. Wyniki analizy wskazują, że zastosowanie w konstrukcji ramowej doczołowych węzłów śrubowych z blachą wystająca o grubości większej niż 15 mm prowadzi do wytworzenia w konstrukcji mechanizmu cięgnowego i zatrzymania katastrofy postępującej.

Received: 2022-03-22, Revised: 2022-04-12