Analysis of steel bolted end-plate joints under accidental situation

D. Kukla & A. Kozłowski

Rzeszow University of Technology, Rzeszów, Poland

ABSTRACT: Shaping and performing building structures analysis based on avoiding the disproportionate damage to the initial cause during accidental situation is nowadays essential concern of designers. A several ways to protect the structures against progressive collapse are presented in current design standards. As a main strategy to mitigate a progressive collapse an alternative load path method is recommended. The ability of structure to overcome collapse is defining as structural robustness. The paper presents results of the part of research project aiming robustness behavior analysis of steel frames with bolted end-plate joints. First step was experimental test of selected, isolated flush and extended end plate joints, to obtain available rotation capacity of the joints tested in column loss scenario. Next, advanced FEM models were created, which after hierarchical validation, were used to simulate experimentally tested joints to compare results. FEM was also used to modeling behavior of subframe tested early experimentally. Last part will be FEM analysis of few plane frames configuration to obtain required rotation capacity of the joints.

1 INTRODUCTION

The subject of numerical simulation of the behavior and evaluation of the resistance of steel frame structures under accidental situation e.g. column loss scenario is a new and intensively developed issue.

The following methodology was applied in PhD dissertation of the first author, to analyze robustness behavior of steel frames subjected to accidental loading:

- Experimental tests of selected bolted joints, i.e. flush and extended end-plate joints. Joints were tested as isolated, in cruciform stand-up.
- Creation and hierarchical validation of FEM model of bolted connections.
- FEM modeling of experimentally tested joints,
- Parametric study on influence of another factors, not including in experimental tests,
- FEM analysis of substructure experimentally tested earlier,
- Analysis of few plane steel frames under column loss scenario.
- Comparison of available rotation capacity from own experimental tests with required rotation capacity obtained from frame analysis,
- Final conclusions about usefulness of bolted end-plate joint in frame structures to fulfill code requirements about structural integrity.

In this paper results of the part of research project described above has been presented.

2 ANALYSIS OF STEEL JOINTS

As a first step to analysis the robustness of frame structures the experimental tests on steel joints were conducted. 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 (Kozłowski & Kukla 2019). Based on these tests the numerical analysis and validation of finite element models of this joints were done, which were presented in (Kukla & Kozłowski & Siwowski 2021). A close agreement of both technique at level of results and graphic comparison (Figure 1) were obtained. The experimental tests and numerical analysis gave a lot of information about behavior of steel end plate joints under accidental situation as column loss scenario.

The validated models of end plate joints were used to conduct the parametric numerical analysis under different factor influencing rotational and ultimate capacity. The three innovative conceptions of bolted end-plate joints to increase load and rotational capacity were also proposed. The results of this analysis were presented in (Kukla& Kozłowski 2021).

3 ANALYSIS OF FRAME SUBSTRUCTURE

The next step to modeling steel frames was numerical analyzes of the frame subsystem (Figure 2) investigated as part of the research work (Kozłowski & et al. 2011). The analysis was 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 end-plate 10 mm thick. Both systems were modeled in the same way.

For this purpose, the analysis of this system (Figure 2) was started in the Abaqus software (Abaqus). Explicit dynamic integration method to overcome convergence difficulties in static solver was 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

Figure 1. Comparison views of experimental and numerical test of joint with 10 mm flush end plate.

Figure 2. Planar view of frame substructure tested in (Kozłowski & Gizejowski & Slecza & ..., 2011).

Figure 3. 3D view of substructure with extended end-plate joint: a) model, b) specimen under test.

(Figure 3a) composed of two elements types was modeled. A parts with the highest values of strains as end-plates, end parts of beams, columns, bolts connectors and pin connections at the end of system were modeled by solid elements C3D8R. The parts of beams between external and internal column (at part about 5 m) were modeled by shell elements S4R. Application of shell element at long beam elements allows to significant reduction of element numbers and computational time of analysis. Both shell and solid elements allows to large displacements and strains at analysis time. To reproduce contact between elements a general contact interaction was developed. 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 (Figure 3b) were applied. The “coupling” option has been used to connect a solid element to a shell element using a reference point.

Three steps of loading procedure were used to analysis. In the first step the axial compression force 10 kN was applied into bolts to simulate initial contact between column flanges and end- plates. The gravity load of whole model as second step was applied to reduce dynamic effects and archiving 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. IPE 300 was used to modeling beams. The flush and extended end-plate with 10 mm thickness was applied as two types of end-plate. The welds at whole model were neglected. M20 bolts were used to connect the elements of the joints. A bolt, washer and nut were modeled as parts of connector. The detail of simulation of bolt connector was presented in (Kukla & Kozłowski & Siwowski 2021).

Corresponding material models were used for each of the model elements. The material properties of HEB200 and IPE 300 were taken from (Kozłowski & et al. 2011). Due to lack of material models of end-plate and bolts, values obtained from own tests (Kozłowski & Kukla

2019) were assumed. The true stress-strain relationship was defined to simulate realistic behavior on material models. The material model used to analysis subframe as in (Kukla & Kozłowski & Siwowski 2021) were adopted.

For solid elements the hexagonal element shape in the mesh control was used. To obtain regular shapes of mesh in all solid elements of model the sweep technique with medial axis algorithm was applied. To fracture simulations the element deletion were employed. The columns, ends of beams and end-plate by 10 mm and 5 mm mesh size were meshed (Figure 4), respectively. The bolt connector was meshed singly. As a main density equal 3,5 mm for bolts and nut were assumed. Quad-dominated element shape with free technique in medial axis algorithm was developed for shell elements. As a size (Figure 4) of mesh 50 mm density was assumed.

Relevant results of the experimental tests of the frame subsystem with bolted joints with a 10 mm thick flush and extended end-plate with numerical analysis results based on the finite element method were compared. The basic behavior curve of the subsystem vertical load - vertical displacement was assessed for compliance of both methods. The force-displacement curve obtained from the experimental and numerical tests is shown in Figure 5a. It can be noticed that up to a specimen deflection of about 200 mm the compliance was almost perfect. After exceeding the displacement of 200 mm, there was a slight increase in the difference of both results. Final displacement upon failure achieved good agreement in both cases. The presented results of the FEM analysis can therefore be considered as a good agreement with the results of experimental studies.

The response curves of the steel subsystem with extended end-plate joints are shown in Figure 5b. A very good convergence was obtained until the vertical displacement of the column was about 100 mm. After exceeding the deflection of 100 mm, the values began to

Figure 4. Detailed view of flush and extended end-plate joint with mesh.

Figure 5. Curve load-vertical displacement of specimen with: a) flush end-plate, b) extended end-plate.

differ slightly. The difference in the final displacement on failure was about 150 mm. The obtained values of the test results and the analysis can be considered acceptable.

Presented analysis confirms that created FEM model can be used in frame structures analysis mentioned in point 4.

The largest deformations of the system were located at the joints (Figure 6). The inner joint of the outermost column was significantly deformed. Large deformations (Figure 6a) can be seen especially in the end-plate. The bend of the column flange was also visible. The comparative image of the deformation of the extreme column shows their good agreement.

The deformation of the middle column joint is presented in the Figure 6b. The significant deformation of the joint (Figure 6b) was obtained in the numerical analysis. The comparison (Figure 6b) of the joint views after the tests confirms the good agreement. Both joints were destroyed by breaking the end plate at the level of the beam bottom flange. The comparison of whole view specimen and model after test was presented in Figure 7.

Figure 6. Failure mode of 10 mm extended end plate: a) external column joint, b) internal column joint.

Figure 7. Failure mode of subframe: a) model, b) specimen.

Figure 8. Static diagrams of analyzed frames: a) two span, b) three span, c) four span.

4 FRAME STRUCTURE ANALYSIS

The numerical dynamic study on progressive collapse of planar steel frames under sudden column loss were presented in (Znang & Jiang & Xu & Wang 2018). Finite element models with refined shell elements were developed. The welded beam-to-column joints were analyzed. It was summarized that ascrucial load and robustness of frame structure in a strong beam-weak column is larger than in a strong column-weak beam ones.

A joint model for assessing the rotational stiffness of semi-rigid connection were adopted in (Li & Li & Jiang & Lu 2018) in the form of multi-spring component model (MSC). The effect of semi-rigid connection as crucial for the deformation and load-distribution were noticed. The results show that the side column loss leads to the largest maximum displacement at the top of the removed column.

The analysis of steel moment frame robustness under vehicle impact was presented in (Kang & Kim 2014). A various speed of vehicle was adapted to assessing structural response. It was found out that effect of collision to the corner column was much greater than collision to the exterior column.

The study of progressive collapse of planar steel frames under column loss were presented in (Klakojour & Sheidaii 2018). The influence of finite element modeling and analysis technique were analyzed. As a major influence on structural response a mesh size and removal time of column were obtained.

The next step of conducted research was to analyze behavior of steel frames presented in Figure 8 in column loss scenario. The models of frames were created at the same way as for subframe. The solid and shell elements were used. The sudden and gradual remove of column support were applied. The results of this analysis will be presented in the next papers.

5 CONCLUSIONS

Whole research project include multi steps process of experimental tests and numerical analyzes to assess the structural robustness under accidental situations of steel planar frame structures with bolted end-plate joints. In detail the project contain experimental tests of isolated bolted end-plate joints, numerical and parametrical analysis of joints, analysis of subframes structures and finally analysis of whole frame planar structures. In this paper results of the part of research project concerning finite element analysis used for a comparative analysis of the results of frame substructure has been presented. Based on obtained first results, the following remarks can be drawn:

- FEM models of experimentally tested bolted end-plate joints, tested in isolation and in sub-frame configuration, were positively checked by the presented comparative analysis of selected results. Presented modeling technique will be usedfor further numerical analysis of the whole frames,

- the application of end plate joint in frame structure leads to obtain a significant load capacity, rotational capacity and large deformation of joint and whole models,
- the advanced FEM modeling with solid and shell elements allows to observe the behavior of all joint components under time analysis, state the weakest components and failure mode of joint and structure.

ACKNOWLEDGEMENT

This research was supported by a Polish Ministry of Science and Higher Education grant to maintain research potential.

REFERENCES

- Dassault, Abaqus – User Analysis User’s Manual. Dassault Systems Simulia Corp.
- Kang, H. & Kim, J. 2014. Progressive collapse of steel moment frames subjected to vehicle impact. *American Society of Civil Engineers*.
- Klakojour, F. & Sheidaii, M.R. 2018. Effects of finite modeling and analysis techniques on response of steel moment-resisting frame in dynamic column removal scenarios. *Asian Journal of civil Engineering*.
- Kozłowski, A. & Gizejowski, M. & Slecza, L. & Pisarek, Z. & Saleh, B. 2011. Experimental investigation of the joints behaviour–Robustness assessment of steel and steel concrete composite frames. *Proceeding of the 6th European Conference on Steel and Composite Structures*: 339–344. Budapest.
- Kozłowski, A. & Kukla, D. 2019. Experimental tests of steel unstiffened double side joints with flush and extended end plate. *Archives of Civil Engineering*. vol. LXV. ISSUE 4:127–154.
- Kukla, D. & Kozłowski, A. 2021. Parametric study of steel flush and extended end-plate joints under column loss scenario. *Engineering Structures* (Paper under review).
- Kukla, D. & Kozłowski, A. & Siwowski, T. 2021. Numerical analysis of steel double side joints with flush and extended end plate under accidental situation. *Eurosteel 2021* (Paper finally accepted).
- Li, L-L. & Li, G-Q. & Jiang, B. & Lu, Y. 2018. Analysis of robustness of steel frames against progressive collapse, *Journal of Constructional Steel Research* 143:264–278.
- Znang, J. & Jiang, J. & Xu, S. & Wang, Z. 2018. An investigation of the effect of semi-rigid connections on sudden column removal in steel frames. *Structures* 13:166–177.