FINITE ELEMENT MODELING OF COLD-FORMED STEEL BOLTED MOMENT CONNECTION

Muhamad F. Akbari[a], Data Iranata[b*], Djoko Irawan[b]

Abstract: *This paper describes the finite element procedure for modelling cold-formed steel bolted moment connection to simulate hysteretic moment-rotation behavior and failure mode. The connection element consists of CFS curved flange beams, double-lipped channel columns, and trough plates. Abaqus software is used in this paper. The modeling procedure includes material properties, bolt modeling, boundary conditions, mesh, loading, and geometrical imperfections. The results of the finite element modeling were compared with the experimental test results in the form of moment-rotation curve and a comparison of failure deformation. It was found that the finite element results had fairly good accuracy in predicting the hysteretic moment–rotation behavior. In the elastic region, the result shows that the finite element model successfully simulates the initial stiffness of the referenced beam-column connection. Meanwhile, the peak moment of the finite element model occurs at the same rotation as the experimental test but the magnitude of the peak moment is lower than the experimental result, which indicates that the finite element model produces a more conservative design. The comparison of failure deformation between finite element model and experimental test shows a very good agreement. The numerical model can simulate well the rotational behavior of the beam-column connection and can predict the general shape and location of local/distortional buckling at the beam-column connection.*

Keywords: *Cold-formed steel, bolted moment connection, finite element method, stiffener, backbone curve*

Submitted: 15 April 2022; Revised: 14 June 2022; Accepted: 14 June 2022

INTRODUCTION

Material innovation in the world of civil engineering has developed quite rapidly, leading to eco-friendly construction, rapid execution, ease of transport, and anti-termites. Other criteria that must be met are consistent in strength and dimensions. Cold-formed steel (CFS) is one of the suitable materials to meet these needs and it has been developed as an alternative to hot-rolled steel and wood. The yield stress of cold-formed steel ranges from 250 MPa to 550 MPa [1]. The thickness ranges from 0.1 mm to 7.9 mm [2] so cold-formed steel sections have the highest strength-to-weight ratio of any material. However, because of the thin material, cold-rolled steel elements are susceptible to local, distortional and global buckling [3]. Due to its low buckling capacity, cold-formed steel is considered more economical for low-rise buildings when the span is not too large [4]. So the material has the potential to produce better and more efficient construction than hot-rolled steel.

A good understanding of the behavior of structural elements is very important for the usage development of cold-formed steel in a country, especially an earthquake-prone country. One of them is the beam-column connection element. The beam-column connection is an important element in the structure that plays a role in transmitting the forces from the beam to the column, supporting the integrity and stability of the overall structural frame system. Understanding the behavior of beam-column connection under cyclic loads is important to know the characteristic of the connection, e.g., joint capacity, joint stiffness, hysteretic moment-rotation behavior, failure mode, ductility, and energy dissipation to be used in the design process. Experimental studies have been carried out to determine the behavior of the beam-column joints of cold-formed steel [5]. However, experimental studies have several constraints such as costs, labor, and testing instruments, so an alternative that can be done is to conduct numerical studies using finite element method. One of the popular finite element software is Abaqus. According to [6] [7], Abaqus was able to capture realistic behavior of cold-formed steel beam-column connection and match the results of the experimental test. So this study will discuss the finite element procedure for modeling cold-formed steel beam-column joints in Abaqus software to determine its behavior.

RESEARCH SIGNIFICANCE

This paper describes the finite element procedure for modeling cold-formed steel bolted moment connection to simulate hysteretic moment-rotation behavior and failure mode by using Abaqus software.

METHODOLOGY

The finite element model is made concerning the experimental research conducted by [5]. In that study, six specimens were made namely A1, A2, A3, B1, B2, and B3 (Table 1). The specimen used as validation in this study is specimen A1, the dimension and configuration can be seen in Figure 1. Specimen A1 was chosen because the failure pattern was dominated by flexural and local buckling, and the connection slip that occurred was small. Model of specimen A1 will be made to evaluate the accuracy of the finite element model in predicting the behavior of cold-formed steel beam-column connection under cyclic loading. The specimen is subjected to cyclic load at the end of the beam which refers to the loading protocol of AISC code section S6.2 [8] about the cyclic qualification of moment resisting connections in special and intermediate moment frames, cyclic loading protocol can be seen in Figure 2. The experimental test setup can be seen in Figure 3. The finite element model was made using the Abaqus 2020 version.

[a]Master Student in the Civil Engineering Department, Institut Teknologi Sepuluh Nopember, ITS Campus, Sukolilo, Surabaya 60111, Indonesia.
[b]Lecturer in the Civil Engineering Department, Institut Teknologi Sepuluh Nopember, ITS Campus, Sukolilo, Surabaya 60111, Indonesia. Corresponding author email address: iranata80@gmail.com/ data@ce.its.ac.id

The material used in the experimental test for the beam, column, and trough plate elements is S275, from the tensile coupon test the yield stress was 310 MPa. Young's modulus and Poisson ration values of cold-formed steel used in the modeling are 210000 MPa and poison ratio 0.33, respectively. The stress-strain curve model used in this study refers to the stress-strain characteristics obtained from tensile coupon tests on steel plates conducted by [9] for specimen A as shown in Figure 4. The stress-strain curves are digitized then the stress and strain values are inputted into Abaqus. Since the reference bolted-moment connection undergoes plastic deformation with strain reversals due to cyclic loading, so the effect of cyclic strain hardening is taken into account in this study. The material model used is combined hardening law in Abaqus.

Table 1 Test specimen in the experimental study by [5]

Specimens	Beam thickness (mm)	Connection stiffeners	Connection type
A1	3	No stiffeners	Slip-critical
A2	3	Partial (minimum) stiffener	Slip-critical
A3	3	Full (optimum) stiffeners	Slip-critical
B1	4	No stiffeners	Slip-critical
B2	4	Partial (minimum) stiffener	Slip-critical
B3	4	Full (optimum) stiffeners	Slip-critical

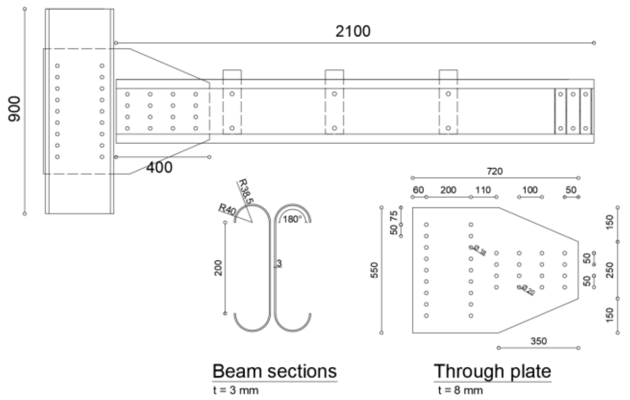


Figure 1 Dimensions and configuration of the test specimen A1 [5]

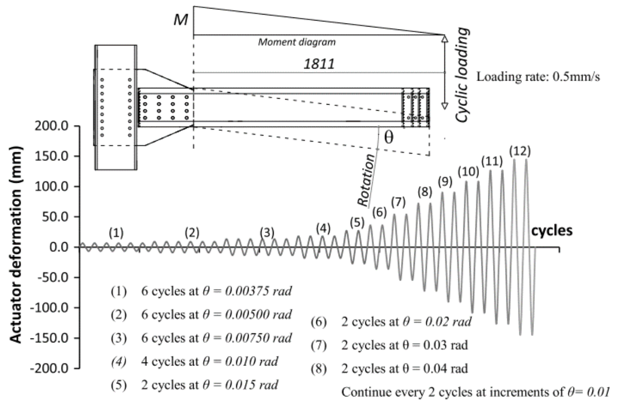


Figure 2 Cyclic loading protocol [5]

In this study, each component of the cold-formed steel connection is modeled in 3D as a shell element using the S8R element type (8-node quadrilateral shell element with reduced integration) found in the ABAQUS library, this model is considered more accurate in predicting hysteretic behavior and energy dissipation capacity of the connection [10]. In addition, based on [9], using shell elements can capture the local instability that occurs. After conducting mesh sensitivity analysis with several mesh sizes, the mesh size 20 mm x 20 mm is selected because it is considered quite accurate and computationally efficient for modeling cold-formed steel connections. Boundary conditions between components defined in this model refer to the experimental test setup by Sabbagh et al [5]. Column stiffeners are connected to the column surface using a "Tie" constraint, column stiffeners are used to keep the column to remain elastic, and the modeling of column stiffeners can affect the rotational response of the connection [11]. It should be noted that in this study, it is assumed that the panel zone remains elastic. In the experimental test setup, the two beams are connected by a bolted infill plate, this plate is modeled with a “Tie” constraint so that the two-beam faces can be connected and coupled together in the U_X, U_Y, U_Z directions. The lateral support in the test setup that prevents out-of-plane deformation is modeled by restrained deformation in the X-direction. For modeling the loading at the end of the beam, the nodes at the end of the beam are attached at the section centroid using coupling constraints. To prevent stress concentration at the end of the beam, the beam stiffener is applied. The beam stiffener is connected to the surface of the beam using a "Tie" constraint. To model the support of the column, the top of the column is restrained in the U_x and U_y directions while the bottom of the column is restrained in the U_x, U_y, and U_z directions. The boundary condition modeling on Abaqus can be seen in Figure 5.

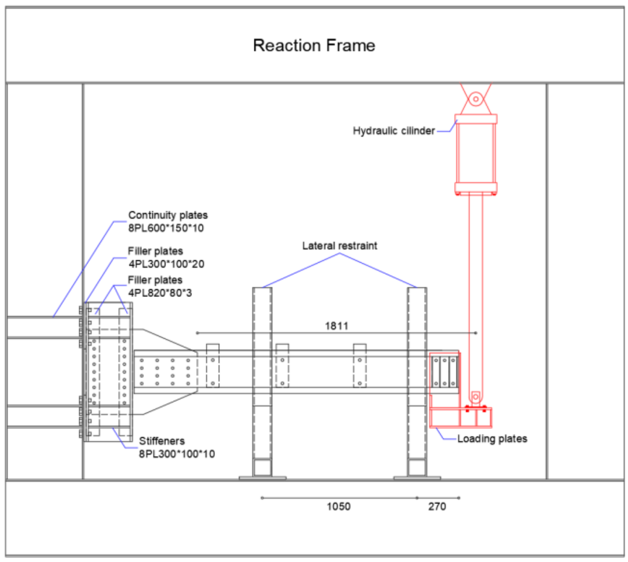


Figure 3 Experimental test setup [5]

In this study, each bolt is modeled using point-based fasteners found in the Abaqus library. The configuration used is two layers of fasteners, each layer connecting the beam to the gusset plate or the column to the gusset plate. Each beam or column node is connected to a CFS plate with a connector element that couples the displacement and rotation of the bolt. To model bolts with point-based fasteners, the bolt radius needs to be defined in terms of a "physical radius" that will represent the bolt radius and simulate the interaction between the bolt and the bolt hole surface. In this study, the rigid behavior of the bolts will be assigned to each bolt. Bolt modeling using point-based fasteners was chosen because it is simpler and more computationally efficient, especially for cyclic loading. It is different if the bolts are modeled as a solid element, it will be more complex and computationally inefficient in cyclic loading, especially if there are many bolts to be modeled. In addition, bolt modeling using point-based fasteners reduce the risk of non-convergent Abaqus calculations.

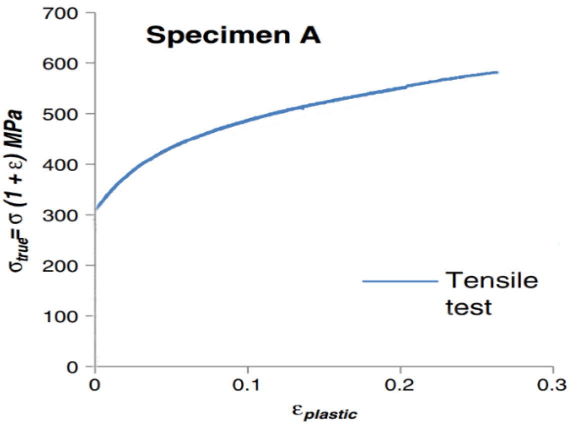


Figure 4 The stress-strain curve used in the model [9]

Based on [12], geometrical imperfection is one of the factors that can affect the behavior of the beam which causes a decrease in strength either under the influence of monotonic or cyclic loads, so geometrical imperfections need to be taken into account in the analysis process. As previously explained, in the test setup, the CFS beam-column connection is restrained by lateral bracing so that global buckling can be ignored and only local and distortional buckling geometrical imperfections are taken into account. Then from both of them, the mode that has the smallest buckling resistance is selected and it will be used in the main model. The values of local and distortional buckling geometrical imperfections for steel sheets with thickness less than or equal to 3 mm are 0.94t and 0.34t, respectively, as recommended by Schafer and Pekoz [13]. Geometric imperfections from local and distortional buckling can be obtained from the first buckling mode by performing an eigenvalue buckling analysis on the CFS beam-column connection element. For monotonic analysis, it is carried out by providing a tip displacement load in the vertical direction (Y) which causes an asymmetrical displacement as shown in Figure 6. For the cyclic analysis, the asymmetrical mode is obtained by combining the results of the monotonic analysis in the +Y and -Y directions.

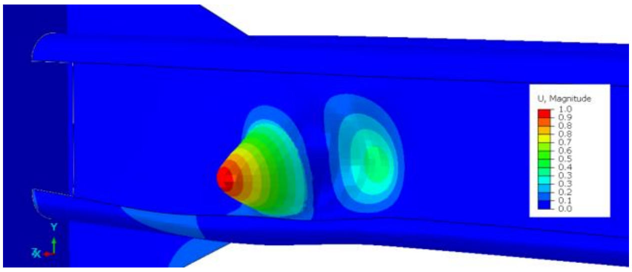


Figure 6 Geometry imperfection of finite element model

ANALYSIS AND DISCUSSIONS

The comparison between experimental and numerical results is presented in the form of moment-rotation curve as shown in Figure 7, the moment that occurs is normalized to the plastic moment of the beam (M_p), where the M_p value is 75 kNm using actual yield stress of 310 MPa according to [5]. The calculated moment (M) and rotation (θ) are taken at the end of the connection according to Figure 2. It can be seen that the numerical simulation results show a good agreement with the experimental test results. In the elastic region, the numerical results compare very well with the results recorded in the experimental test. This shows that the finite element model successfully simulates the

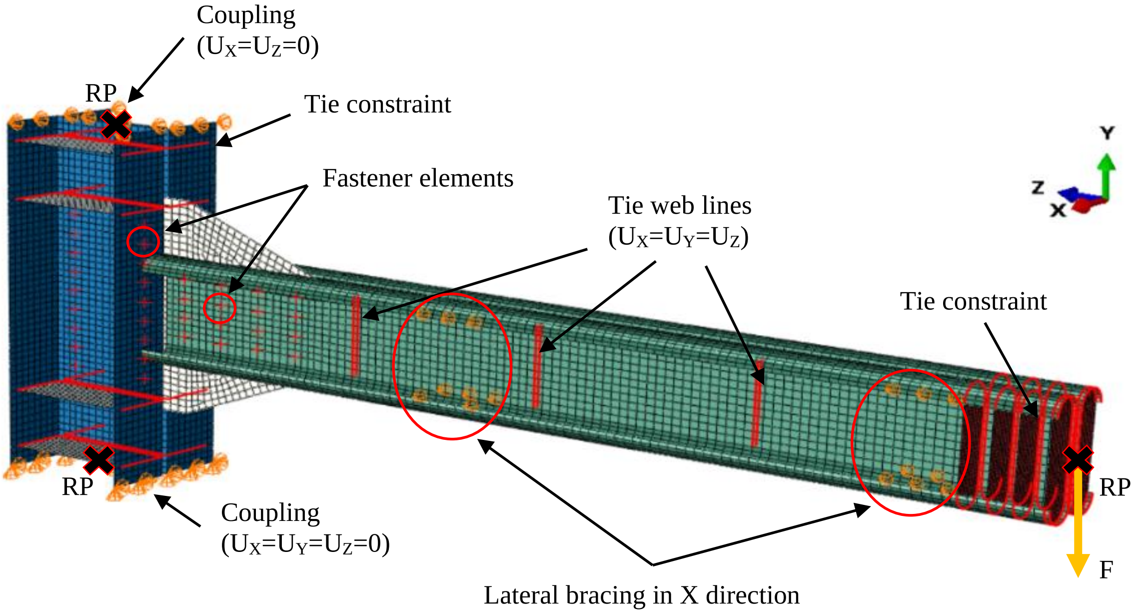


Figure 5 Boundary condition of finite element model for beam-column connection

initial stiffness of the referenced beam-column connection. From Figure 7, it can be observed that the peak moment of the finite element model occurs at θ = 0.02 rad with the magnitude of moment is 0.79Mp, while in the experimental results the peak moment occurs at θ = 0.02 rad with the magnitude of the moment is 0.83M_p.

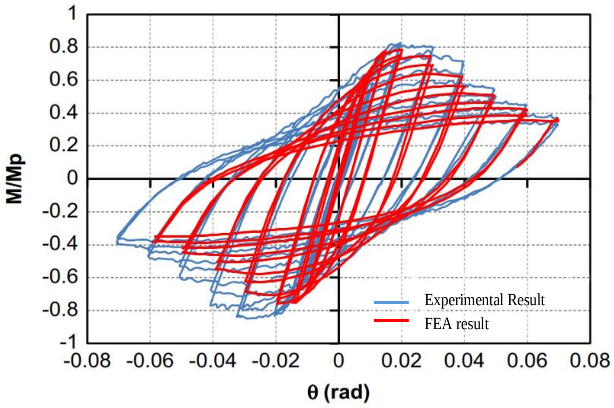


Figure 7 Backbone curve comparison between numerical models and experimental tests

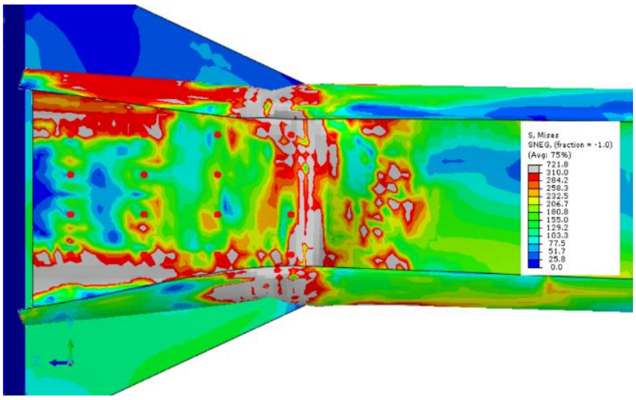
Although the peak moment of the finite element model occurs at the same rotation as the experimental test, there is a 4.8% difference in the peak moment magnitude. This can be caused by differences in geometric imperfections and material properties between the finite element model and the experimental specimen. The lower peak moment of the finite element model can indicate that the finite element model produces a more conservative design. After the peak moment occurs, the buckling in the beam increases causing a continuous degradation of stiffness leading to a decrease in the moment capacity until the loading phase ends. However, it can be observed in Figure 7 that the numerical model cannot reach θ = -0.07 rad as the experimental test results. This is caused by the divergence of numerical analysis. This happens because the crack is initiated so that the numerical analysis fails to converge and the hysteresis behavior of the connection cannot be caught. In fact, the numerical divergence does not always indicate the actual failure point. The difference between the results of numerical and experimental analysis can be interpreted as residual strength, so the numerical model produces a more conservative design.

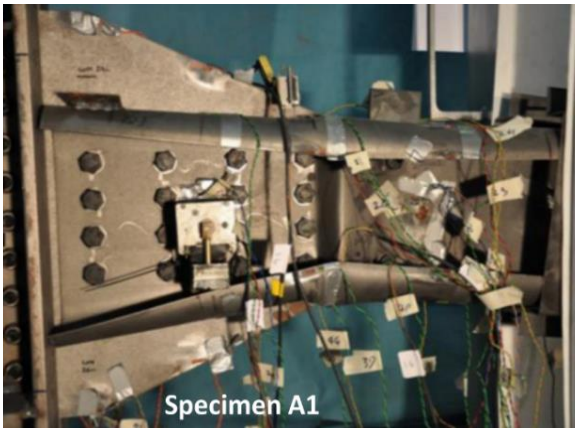
Besides moment-rotation comparison, for better comparison, a comparison of failure deformation between finite element model and experimental test is carried out (Figure 8). The comparison of failure deformation is taken at the end of the cyclic loading where the rotation reaches 0.069 rad. It can be seen that the numerical simulation results show a good agreement with the experimental test results. From the results of the finite element analysis, it was found that the rotational behavior of the beam-column connection was dominated by flexural and local buckling deformations in the beam, this is in accordance with the rotational behavior captured from experimental tests. Besides that, the failure deformation from the finite element analysis is similar to that recorded in experimental tests, where there is buckling in the flange and beam body in the connection area which causes a sudden loss of strength, the beam flange opens as the displacement increases. Thus, the numerical model can simulate well the rotational behavior of the beam-column connection and can predict the general shape and location of local/distortional buckling at the beam-column connection. In addition, from Figure 8a can be observed the distribution of the von-Mises stress at the connection, where the gray area is the area that is experiencing yielding. Besides, the stress that occurs in the gusset plate is smaller than the beam, this is because the gusset plate is thicker.

CONCLUSIONS

A numerical investigation was carried out to model the cyclic behavior of the cold-formed steel bolted moment connection. The numerical analysis results were then compared with the results of the experimental tests to test the validity. The finite element software namely Abaqus is used in the numerical analysis process. Abaqus modelling steps have been presented in this literature.

The comparison between experimental and numerical results is presented in the form of a backbone curve of the moment-rotation and the comparison of failure deformation. The numerical simulation results show a very good agreement with the experimental test results. In the elastic region, the numerical results compare very well with the results recorded in the experimental test. This shows that the finite element model successfully simulates the initial stiffness of the referenced beam-column connection. The peak moment of the finite element model occurs at the





(a) (b)

Figure 8 Comparison of failure patterns (a) numerical model (b) experimental test same rotation (θ = 0.02 rad) as the experimental test, there is a 4.8% difference in the peak moment magnitude, where the peak moment of the numerical model is $0.79M_p$, while in the experimental results the peak moment is $0.83M_p$. The lower peak moment of the finite element model can indicate that the finite element model produces a more conservative design. The comparison of failure deformation between the finite element model and the experimental test shows a very good agreement. The numerical model can simulate well the rotational behavior of the beam-column connection and can predict the general shape and location of local/distortional buckling at the beam-column connection.

The use of numerical investigations is very beneficial in determining the behavior of an element, not least cold-formed beam-column connection, which will be very important in the development of cold-formed steel materials in a country that rarely uses cold-formed steel as the main structural element.

REFERENCES

[1] G. J. Hancock, "Light gauge construction," Progress in Structural Engineering and Materials, vol. 1, no. 1, pp. 25-30, 1997.

[2] W.-W. Yu, V. A. S. Liu, and W. M. McKinney, "Structural behavior of thick cold-formed steel members," Journal of the Structural Division, vol. 100, no. 11, pp. 2191–2204, 1974.

[3] M. M. Billah, R. Islam, and A. Bin, "Cold formed steel structure: an overview," World Scientific News: An International Scientific Journal, vol. 118, pp. 59-73, 2019.

[4] O. Tunca, F. Erdal, A. E. Sağsöz, and S. Çarbaş, "Structural features of cold-formed steel profiles," Challenge Journal of Structural Mechanics, vol. 4, no. 2, p. 77, 2018.

[5] A. B. Sabbagh, M. Petkovski, K. Pilakoutas, and R. Mirghaderi, "Experimental work on cold-formed steel elements for earthquake resilient moment frame buildings," Engineering Structures, vol. 42, pp. 371–386, 2012.

[6] J. Ye, S. M. Mojtabaei, and I. Hajirasouliha, "Seismic performance of cold-formed steel bolted moment connections with bolting friction-slip mechanism," Journal of Constructional Steel Research, vol. 156, pp. 122–136, 2019.

[7] A. B. Sabbagh, M. Petkovski, K. Pilakoutas, and R. Mirghaderi, "Development of cold-formed steel elements for earthquake resistant moment frame buildings," Thin-Walled Structures, vol. 53, pp. 99–108, 2012.

[8] A. I. S. Construction, Seismic Provisions for Structural Steel Buildings Supersedes the Seismic Provisions for Structural Steel Buildings, Chicago, 2005.

[9] A. B. Sabbagh, M. Petkovski, K. Pilakoutas, and R. Mirghaderi, "Cyclic behaviour of bolted cold-formed steel moment connections: FE modelling including slip," Journal of Constructional Steel Research, vol. 80, pp. 100–108, 2013.

[10] P. Foster, G. Abdelal, J. B. P. Lim, M. Hajsadeghi, and D. McCrum, "Finite element modelling of cyclic behaviour of cold-formed steel bolted moment-resisting connections," Journal of Construction in Developing Countries, vol. 21, no. 1, pp. 167–180, 2016.

[11] R. Tartaglia, M. D'Aniello, G. A. Rassati, J. A. Swanson, and R. Landolfo, "Full strength extended stiffened end-plate joints: AISC vs recent European design criteria," Engineering Structures, vol. 159, pp. 155–171, 2018,

[12] M. D'Aniello, R. Tartaglia, S. Costanzo, and R. Landolfo, "Seismic design of extended stiffened end-plate joints in the framework of Eurocodes," Journal of Constructional Steel Research, vol. 128, pp. 512–527, 2017.

[13] B. W. Schafer and T. Peköz, "Computational modeling of cold-formed steel: characterizing geometric imperfections and residual stresses," Journal of Constructional Steel Research, vol. 47, no. 3, pp. 193-210, 1998.