Numerical analyses of a base connection for a thin-walled cold-formed profile column

V M Venghiac1, I Olteanu-Donțov1, M S Alexa-Stratulat1, G Țăranu1, S A Băetu1, I O Toma1
1 The “Gheorghe Asachi” Technical University of Iasi, Faculty of Civil Engineering and Building Services, Department of Structural Mechanics, no. 1, Prof.dr.doc. D. Mangeron Street, 700050, Iasi, Romania
E-mail: ionut.ovidiu.toma@tuiasi.ro

Abstract. Thin-walled cold-formed steel structures tend to gain more attention in the civil engineering field. Ground-floor dwellings, as well as factory warehouses represent the most frequent use of these structural systems. However, due to the inherent flexibility of thin-walled cold-formed steel elements, the designer has to take into consideration the possibility of large deformations occurrence, especially in the case of structures with large openings. Such deformations may lead to significant stresses at the level of the connection joints between elements or between elements and the foundation block. The paper presents a comparative numerical analysis based on the finite element method to assess the efficiency of a connection joint between the columns of a portal frame and the foundation block. The linear and nonlinear analyses were carried out by means of Robot Structural Analysis and Ansys. The column is made of two thin-walled cold-formed C shape profiles positioned back-to-back. The efficiency was assessed both from a stress-state point of view in the column and in the connection parts, as well as from the point of view of the manufacturing process and implementation. Significant differences were observed regarding the stress distribution in the nonlinear analysis.

1. Introduction

In the construction industry there are two types of steel profiles commonly used: hot-rolled and cold-formed steel profiles. The differences between them reside in the used technological process, their shapes and mechanical characteristics [1]. The cold-formed steel profiles (CFSP) were developed as a lightweight alternative for the traditional materials. Nowadays, structures made with this solution cover over 45 percent of the construction market and the number is continuously rising [2,3].

CFS elements have a wide range of application in the construction field from non-structural components (e.g. roof purlins, rails for wall cladding, etc.) to load-bearing systems or subsystems (trusses, rack structures, portal frames, moment-resisting frame, floor decking, etc.) [3,4-6].

Even though CFS members began to be used since the 1850s in the US and Great Britain, it was only after the 1920s and 1930s when it was accepted as a construction material, but with limited usage due to lack of research and official codes. The Virginia Baptist Hospital in Lynchburg, Virginia, constructed in 1925 is considered to be among the first buildings that used CFSP as construction elements. The structural system of the building was load bearing masonry with floors made of double back-to-back cold-formed steel lipped channels. The CFSP usage increased significantly after 1946 when the first prescription for this material appeared.

Despite the advantages of CFS elements, such as higher strength per unit weight compared to hot rolled steel elements, ease and speed of construction coupled with a higher flexibility in manufacturing, there are several draw-backs mainly caused by the very small thickness of the steel section which lies between 0.9 mm and 3.2 mm [7]. The local buckling and general instabilities of CFS elements may prevent their application in structures located in active seismic areas prone to severe earthquakes.

Extensive research was conducted to assess the behaviour of structural systems subjected to seismic excitations. The joints between the structural elements or between the elements and the foundation are
sometimes considered to be the weakest link in the whole system and therefore gathered the much-needed attention from researchers worldwide [4,8-12].

The numerical modelling and analysis are one of the most convenient methods for assessing the behaviour of CFS elements and/or structures subjected to a wide range of loading scenarios. The main advantage of numerical modelling over the experimental investigations is the possibility of varying a large number of influencing parameters without the limitations of cost, time and instrumentation of the latter. On the other hand, modelling the behaviour of CFS elements requires careful considerations of geometry, contact and material nonlinearities that may result, if not properly addressed, in convergence issues [13-15].

The paper presents the results obtained from a comparative analysis based on the finite element method (FEM) to assess the efficiency of a connection joint between the columns of a portal frame and the foundation block considering both the linear and nonlinear behaviour of the CFS elements. While linear elastic analysis is the preferred approach in practical design, it lacks the ability to adequately predict the local buckling of CFS elements and should be used with caution. On the other hand, nonlinear FEA, although being much more computationally intensive, manages to render evident such failure modes frequently met in the case of CFS elements.

2. Methodology

2.1. Geometry
The geometry of the considered model is shown in figure 1. The considered assembly consisted of two 150×45×10 mm CFS C-shape lipped profiles positioned back-to-back, having a thickness of 2 mm. The two CFS profiles were connected at the top by means of two M16 bolts via a steel plate. The base of the column was fixed to a gusset plate by means of two rows of 3×M16 bolts located at a distance of 100 mm, as shown in figure 1a.

2.2. Materials
The material properties of the CFS profile, as considered in the computer software, are given in table 1. An idealized stress-strain curve was considered for the material, as presented in figure 2.
Table 1. Material characteristics

| Material | Young’s modulus, E [GPa] | Tangent modulus [GPa] | Shear modulus, G [GPa] | Poisson’s ratio | Yield Strength [MPa] |
|----------|--------------------------|-----------------------|------------------------|----------------|---------------------|
| S235     | 210                      | 140                   | 81                     | 0.2963         | 235                 |

2.3. Numerical model

The numerical model was generated following the geometry shown in figure 1. Two computer programs were used and two types of analyses were considered (static linear and non-linear analyses). The two computer programs were chosen as being representatives for their main field of application: design practice (Robot Structural Analysis [16]) and research (Ansys [17]).

A bilinear model for the material behaviour of steel was considered, as shown in figure 2. The material characteristics were the ones presented in table 1.

While Robot Structural Analysis does not allow the user to choose the type of finite element to be used for mesh generation, Ansys gives the user the freedom to choose between a large library of finite element types suitable for the analysis to be conducted. For both linear and non-linear analyses, a SHELL181 quadratic plane stress element type was considered, figure 3.

Three distinct numerical simulation cases were considered: Case 1 – linear static analysis using Robot Structural Analysis software package, Case 2 – linear static analysis using Ansys and Case 3 – non-linear static analysis using Ansys. The non-linear analysis included the residual stress and imperfections, captures the local and overall buckling behaviour, as well as the local distortions of the connections. The maximum mesh size was considered to be 10 mm for all three cases. A brief characterisation of all modelling scenarios is presented in table 2.

The mesh for the models using the two programs is shown in figures 4 and 5.

Table 2. Characteristics of the modelling scenario

| Model     | Analysis type               | FE type                | Maximum element size (mm) | Number of elements | Number of nodes |
|-----------|------------------------------|------------------------|---------------------------|--------------------|-----------------|
| Case 1    | Linear Robot Structural Analysis | Triangles and squares | 10                        | 7467               | 4096            |
| Case 2    | Linear Ansys                 | Squares, quadrilaterals| 10                        | 10824              | 11201           |
| Case 3    | Nonlinear Ansys              |                        |                           |                    |                 |
a. General view of the model  
b. Detailed view of the meshing  

**Figure 4.** Generated model using Robot Structural Analysis

a. General view of the model  
b. Detailed view of the meshing  

**Figure 5.** Generated model using Ansys 2021R1

The load was applied at the top of the column, in Y direction of the coordinate system, at 975 mm from the horizontal steel plate at the base, figure 1a, in incremental steps, until the failure of the model occurred. For the nonlinear analysis case the failure of the element was considered as the onset of the local buckling or stress levels that exceeded the yield strength of the CFS, according to table 1. The displacements at the base of the steel plate were restrained in all 3 directions, so that to simulate a fixed end. Perfect bond was assumed between the bolts and the connected steel pieces.

The obtained results are analysed from the point of view of stress levels and displacements along the two in-plane directions (X and Y). The analyses will offer some insights on the overall behaviour of the column and the connection at the base and will be further validated and calibrated by means of experimental data obtained from laboratory tests on similar / identical specimens.

3. Results and discussions

3.1. Displacements

The maximum lateral displacements were recorded at the top of the column where the lateral force was applied. As it can be seen from figure 6, both FEM software led to almost the same results considering
the linear analysis: 3.99 mm for Robot Structural Analysis (figure 6a) and 4.08 mm for Ansys (figure 6b), with a difference between the obtained results of 2.25%. However, for the same magnitude of the applied lateral force, the non-linear analysis conducted using Ansys 2021 R1 lead to a maximum value for the displacement at the top of the column of 4.34 mm (figure 6c). The 6.11% difference compared to the results obtained by means of linear elastic analysis could be explained by the redistribution of stresses, considering the material behaviour presented in figure 2, during the deformation process.

Figure 7 shows the distribution of the displacements in X direction, perpendicular to the direction of loading, at the same intensity of the lateral force for which figure 6 was plotted.

It can be observed that the maximum values for the displacements in the X direction were obtained for the points located at the mid height of the column while the top part of the column exhibited 0 lateral displacements with respect to the direction of loading [19]. The latter can be explained by the fact that the movement of the top of the column in X direction, outside the loading plane, is restrained due to the applied force. However, the part of the column with unrestrained displacements in X direction, perpendicular to the direction of loading, is the one located between the top of the gusset plate at the base and the load point of application.

Figure 6. Horizontal displacements in Y direction (direction of loading)

Figure 7. Horizontal displacements in X direction (perpendicular to the direction of loading)
According to the data presented in figures 6 and 7 the magnitude of the displacement in X direction is almost 25% of the displacement in Y direction due to the fact that the element is very flexible. This may lead to the conclusion that the column could have failed due to local buckling of the flanges of the CFS C-shaped lipped profile [20].

3.2. Stresses

Figure 8 presents a comparison in terms of the maximum principal tensile stresses obtained in Robot Structural Analysis and Ansys 2021R1. It can be observed that both software packages estimated a value of the maximum principal stress around 500 MPa, meaning the failure of the column. However, these high values were localized at the level of the third bolt on the tensioned side of the column. This would be an indicator of a local failure mechanism. The stress levels for the rest of the element did not exceed the yield limit.

The intensity of the laterally applied load, 5000N, was identical to the one that produced the displacements in X and Y directions presented in figure 7 and figure 6, respectively. According to the data presented in figures 6a, 7a and 8a, the buckling of the compressed flange of the CFS C-shaped lipped profile was observed. The local failure of the specimen at such a small intensity of the lateral load could be explained by the small thickness of the steel profile, 2mm.

![Figure 8. Principal stress distribution](image)

**Figure 8.** Principal stress distribution

![Figure 9. Principal stress variation for Case 3 at different loading stages](image)

**Figure 9.** Principal stress variation for Case 3 at different loading stages
Figure 9 presents the evolution of the principal stress at different loading stages for Case 3 (table 2). Compared to figure 8, it can be observed that at the same magnitude of the lateral load, 5000N, the value of the principal tensile stress exceeded the yield limit slightly at the same location. The lower values for the principal tensile stress could be attributed to the possibility of stress distribution in the case of nonlinear static analysis compared to the linear one.

Increasing the load to 8500 N results in a larger area of the CFS column being subjected to tensile stresses that exceed the yield strength. The yielded part tended to concentrate on the tensioned side of the column, right above the gusset plate. As the load increased to 10000 N, 15% increase from 8500 N, the maximum value of the principal tensile stress decreased and an even larger area of the CFS profile yielded.

Localized failure could also be observed at the level of bolts located in the compressed part of the column. This would suggest local tearing of the CFS profile if these stresses are analysed taking into account the out-of-loading-plane behaviour presented in figure 6, showing a separation tendency of the two CFS profiles.

4. Conclusion
The paper presents the results obtained from a comparative analysis based on the finite element method (FEM) to assess the efficiency of a connection joint between the columns of a portal frame and the foundation block considering both the linear and nonlinear behaviour of the CFS elements. The column is made of two 150×45×10 mm CFS C-shape lipped profiles positioned back-to-back, having a thickness of 2 mm.

The values of the lateral displacements in the direction of the applied load are similar between the two employed computer programs as well as between the linear and nonlinear analysis cases. The latter is 6.11% larger compared to the linear analysis case. Important out-of-loading-plane displacements are observed for the points located at the mid height of the column. The magnitude of these displacements is almost 25% of the in-plane lateral ones. This means that the two back-to-back CFS profiles exhibit a separation tendency.

The failure of the column initiates at the level of the connection with the gusset plate. More specifically, local yielding is observed at the contact point between the CFS profile and the top bolt in the tensioned part of the column. The nonlinear static analysis is able to capture the redistribution of stresses after initial yielding. Additionally, local buckling of the compressed CFS profile flange is observed.

The model needs validation and calibration via experimental tests but it manages to offer some insights on the expected failure mode of the column and the magnitude of the laterally applied load.

Acknowledgments
The authors wish to express their gratitude to the partnership between the “Gheorghe Asachi” Technical University of Iasi, INAS S.A. and ANSYS for making available the full research license for Ansys 2021R1 used in conducting the research. The authors thank S.C. Panoterm Import-Export S.R.L. for supplying the CFS profiles used in the research as part of the grant no. 4887/03.2019.

5. References
[1] Mojtabaei S M, Kabir M Z, Hajirasouliha I and Kargar M 2018 Analytical and experimental study on the seismic performance of cold-formed steel frames Journal of Construction Steel Research 143 18-31
[2] Schafer B W 2011 Cold-formed steel structures around the world Steel Construction 4(3) 141-9
[3] Smith B H, Arwade S R, Schafer B W and Moen C D 2016 Design component and system reliability in a low rise cold formed steel framed commercial building Engineering Structures 127 434-46
[4] Zeynalian M, Shelley A and Ronagh H R 2016 An experimental study into the capacity of cold-formed steel truss connections Journal of Construction Steel Research 127 176-186
[5] Karabulut B and Soyoz S 2017 Experimental and analytical studies on different configurations of cold-formed steel structures Journal of Construction Steel Research 133 535–46
[6] Roure F, Pekoz T, Somalo M R, Bonada J, Pastor M M and Segura A 2019 Cross-aisle stiffness analysis of industrial welded cold-formed steel rack upright frames Thin-Walled Structures 141 332-44
[7] Lee Y H, Tan C S, Mohammad S, Tahir M M and Shek P N 2014 Review on cold-formed steel connections The Scientific World Journal 2014 ID. 951216
[8] Mathe A E, Catarig A and Moldovan I 2015 Statics and kinematics of semirigid steel frames under seismic action Journal of Applied Engineering Sciences 5(2) 59-64
[9] Campian C M, Pop M and Chira N 2017 The influence of the connections between concrete and steel on the joints modelling ce/papers – Proc. of Eurosteel, September 2017, Copenhagen, Denmark 1(2-3) 4680-7
[10] Bondock D H and Salim H A 2017 Failure capacities of cold-formed steel roof trusses end-connections Thin-Walled Structures 121 57-66
[11] Ye J, Becque J, Hajirasouliha I, Mojtabaei S M and Lim JB P 2018 Development of optimum cold-formed steel sections for maximum energy dissipation in uniaxial bending Engineering Structures 161 55-67
[12] Shahini M, Bagheri Sabbagh A, Davidson P and Mirghaderi R 2019 Development of cold-formed steel moment-resisting connections with bolting friction-slip mechanism for seismic applications Thin-Walled Structures 141 217-31
[13] Bagheri Sabbagh A, Petkovski M, Pilakoutas K and Mirghaderi R 2013 Cyclic behaviour of bolted cold-formed steel moment connections: FE modelling including slip Journal of Construction Steel Research 80 100-8
[14] Rinchen R and Rasmussen K J R 2019 Behaviour and modelling of connections in cold-formed steel single C-section portal frames Thin-Walled Structures 143 ID. 106233
[15] Rinchen R and Rasmussen K J R 2019 Numerical modelling of cold-formed steel single C-section portal frames Journal of Constructional Steel Research 158 148-155
[16] Robot Structural Analysis, https://www.autodesk.com/products/robot-structural-analysis/overview?term=1-YEAR
[17] Ansys 2021 R1, https://www.ansys.com/products/structures/ansys-mechanical
[18] https://www.mm.bme.hu/~gyebro/files/ans_help_v182/ans_elem/Hlp_E_SHELL181.html
[19] Arrayago I, Rasmussen K J R and Real E 2020 Statistical analysis of the material, geometrical and imperfection characteristics of structural stainless steels and members Journal of Constructional Steel Research 175 ID. 106378
[20] Dhanapal J, Ghaednia H, Das S and Velocci J 2020 Behavior of thin-walled beam-column modular connection subject to bending load Thin-Walled Structures 149 ID. 106536