Zeszyty Naukowe Politechniki Częstochowskiej
Budownictwo
ISSN 0526-5916

Zeszyty Naukowe Politechniki Częstochowskiej nr 25 (2019), 92-98
DOI: 10.17512/znb.2019.1.14

Numerical analysis of the stability of a two-story building made using a light steel skeleton frame

Mariusz Kosiń[1]

ABSTRACT:
The article presents a numerical analysis of a single-family house made using the technique of a light steel skeleton frame. Numerical models were prepared for the adopted building, which was subjected to stability analysis using the Robot Structural program. The analysed construction lists possible loads that occur in accordance with applicable standards. In addition, a bending - torsional analysis was performed for the separated part of the external partition. This analysis was performed in the Ansys 18.1.1 Academic Research program. Based on the results obtained, the impact of the considered models on the stability of the building body and the torsion angle of selected elements of the vertical partition were assessed.

KEYWORDS:
light steel skeleton; FEM; thin-walled elements

1. Introduction

The light steel skeleton technique based on thin-walled elements finds application in industrial, single-family housing and extensions of existing buildings. Characteristic for these elements is that the dimension determining the cross-section (thickness) is far thinner when compared to other techniques [1].

This work presents a numerical analysis of a single-family house made using a light steel skeleton frame. The study presented assumes permanent mechanical damage to the skeleton frame of the wall structure, which is made of thin-walled C-profiles. As a result, it becomes necessary to dismantle the stiffening plates to replace damaged parts. In the analysis, three calculation models were assumed.

The purpose of this study is the numerical analysis of the stability of the building's structure assuming the following calculation models: Model 0 - building structure with a solid shell (Fig. 1a), Model 1 - building structure without stiffening plates in the ground floor (Fig. 1b), and Model 2 - building construction without stiffening plates in the ground floor, with vertical bracing (Fig. 1c). Additionally, an analysis was made for the separated part of the external partition. The analyses were carried out in two different FEM programs.

2. Numerical models

2.1. Calculation model of solid building stability

A two-story building measuring 9.3 m by 12.3 m, with a height of 8.62 m and a roof slope of 43° was established for this purpose. The building was designed in accordance with applicable

[1] Czestochowa University of Technology, Faculty of Civil Engineering, ul. Akademicka 3, 42-218 Częstochowa, e-mail: mkosin@bud.pcz.pl, orcid id: 0000-0003-2683-7784

regulations. The adopted model was based on the SundaySystem technology, whose skeleton frame was constructed using 1.5 mm thick C140 sections spaced every 60 cm. In contrast, the floor beams formed integrated C140 profiles reflecting the I-section [2]. The analysed structure lists possible loads that may occur in accordance with applicable standards [3-5].

Calculation models were created in Autodesk Robot Structural 2015. Restraints were adopted as supports, which correspond only partially to real-life conditions. The supporting structure consisted of rod elements which were given releases, thus creating articulated joints. The building structure's sheathing was modeled as panels with OSB parameters.

Figure 1 shows the adopted calculation models and Table 1 the obtained displacement values in the stability analysis.

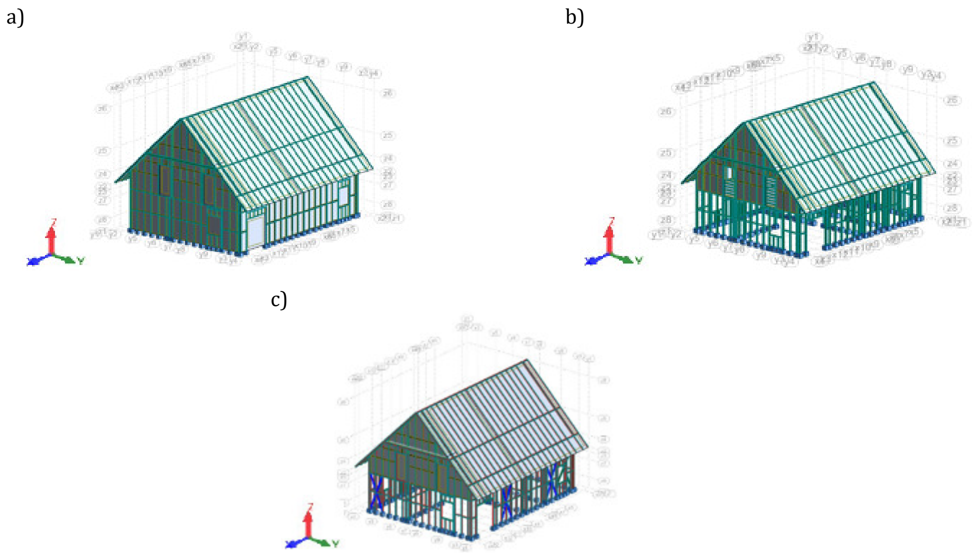

Fig. 1. Calculation models adopted: a) Model I - building structure with all sheathing, b) Model II - building structure without stiffening plated on the ground floor, c) Model III - building structure without stiffening plates on the ground floor, with vertical bracing

Table 1

Displacement results for accepted calculation models

Computational model	Maximum (+) and minimum (–) displacements					
	UX [mm]	UY [mm]	UZ [mm]	RX [rad]	RY [rad]	RZ [rad]
Model I	0.000	3.317	0.000	0.014	0.005	0.001
	–8.994	–1.358	–0.986	–0.004	–0.004	–0.003
Model II	0.000	29.891	0.000	0.025	0.009	0.006
	–230.53	0.000	–0.983	–0.011	–0.081	–0.002
Model III	0.000	10.182	0.000	0.020	0.007	0.002
	–6.291	0.000	–0.967	–0.003	–0.011	–0.004

According to the assumptions, disassembly of the sheathing from the ground floor part would affect the stability of the building body (Table 1) and exceed the limits of horizontal displacements specified according to PN-EN 1993-1-1. On the other hand, the use of vertical bracing in the form of vertical brackets meant that the permissible deviations calculated in accordance with [6, 7] could be maintained.

2.2. Numerical analysis of the separated part of the outer wall

This part of the article presents an analysis of the separated section of the external partition without stiffening plates located on the ground floor of the building (Fig. 2). The selection of the considered section was made on the basis of the maximum displacements obtained from the analysis of the stability of the building structure (point 2.1). The values of the nodal displacements in the selected section of the structure served as boundary conditions in the detailed analysis carried out in the program Ansys 18.1.1 Academic Research.

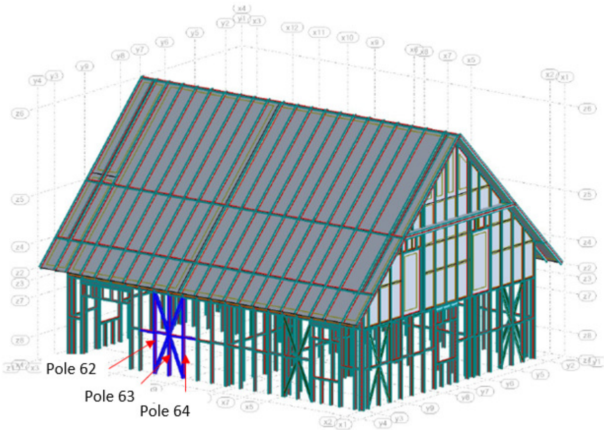

Fig. 2. Computational model of a single-family house construction in Autodesk Robot Structural 2015 with selected profiles for detailed analysis

The detailed analysis included three models - Model 0, Model 1 and Model 3. Model 0 treated as the starting model did not have vertical concentrations (Fig. 3a). Subsequent models were created by supplementing with concentrations and differed in the method of connecting them to the column (Fig. 3b and 3c). Braces in Model 1 were connected by bolts to one of the profile shelves (Fig. 3b) and in Model 2 by two element shelves (Fig. 3c).

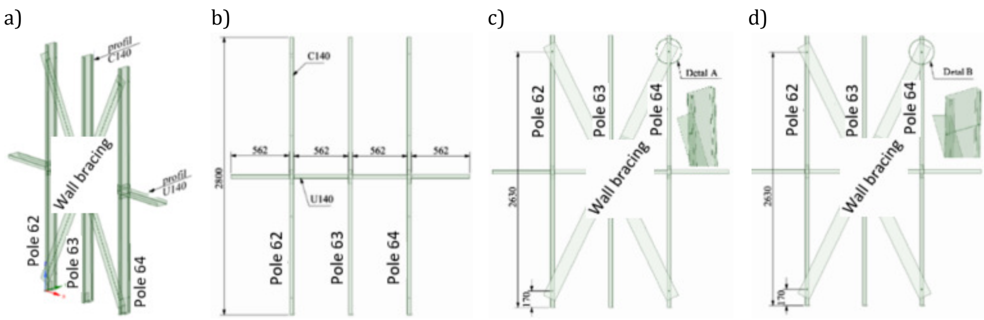

Fig. 3. Diagram of the wall section being analyzed: a) general view of selected profiles, b) Model 0 without bracing, c) Model 1 with bracing attached to one of the profile shelves (detail A), d) Model 2 with bracing attached by two profile shelves (detail B)

Mapping of boundary conditions in ANSYS was possible by assigning displacement readings obtained from the ROBOT Structural program applied at the center of gravity of the cross-section (Fig. 4a). Supports were adopted as reinforcements in places where the partition was connected to the foundation using M10 anchor rods (Fig. 4b).

In the adopted model, Shell181 shell elements for thin-walled elements and Solid187 solid elements for screws were generated. 4-node shell elements in each of the 6 degrees of freedom were used in surface objects. All solid elements have 3 degrees of freedom [8-12]. Two types of contacts were adopted for modeling relations between parts. Bonded contacts, which do not allow a pair of contacts to separate and move, and frictional contact that allows contacts to separate and move each other [9-13].

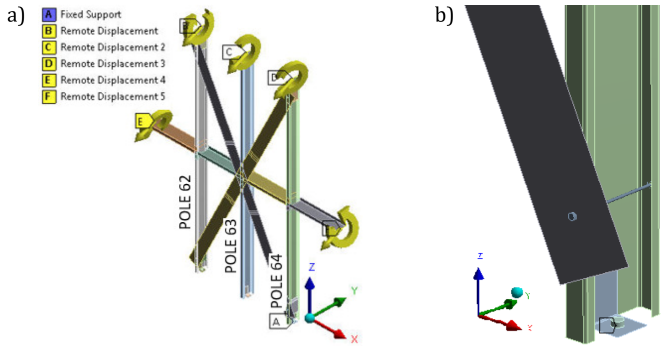

Fig. 4. Boundary conditions of the analyzed section: a) displacements applied at the center of gravity of the cross--section of the profiles, b) support in the form of anchoring the M10 anchor (example of a long bolt connection)

Depending on the considered model, the generated finite element mesh differed in the number of nodes and elements (Table 2).

Table 2
Finite element statistics

Model	Number of nodes	Number of elements
Model 0	168355	164379
Model 1	232535	221360
Model 2	245836	227579

3. Analysis of results

In the considered models, the torsion angle was observed along the web length and on the shelf length for selected columns 62, 63 and 64. Figures 5-10 show the torsion angle distribution in selected elements. The results of the analysis show the influence of wall bracing and the method of their attachment on the torsion angles obtained. The minimum torsion angle is in Model 2 with long screws. For example, for pole 62, the difference between Model 0 and Model 2 is the torsion angle at web height, and at the top of the bracing attachment is approximately 22%. The situation is similar in other cases.

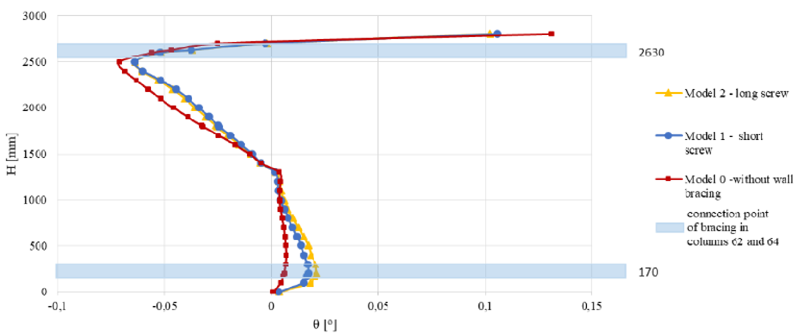

Fig. 5. The course of torsion angle variability along the column web length 62

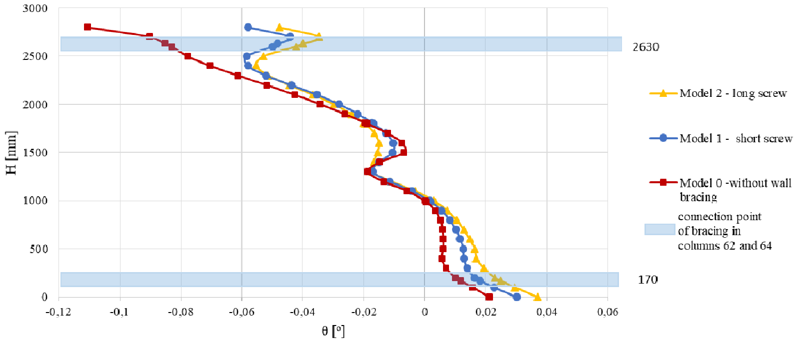

Fig. 6. The course of torsion angle variability along the length of the column shelf 62

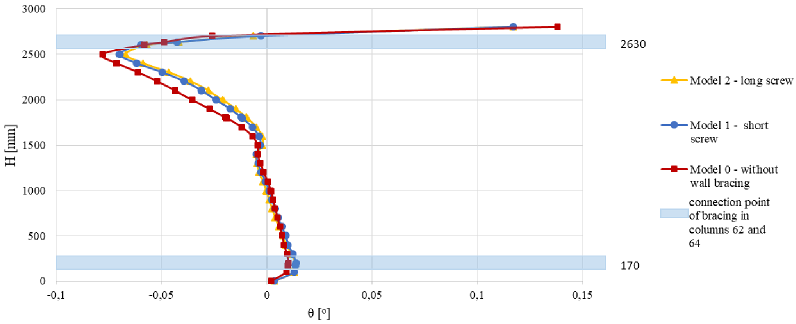

Fig. 7. The course of torsion angle variability along the column web length 63

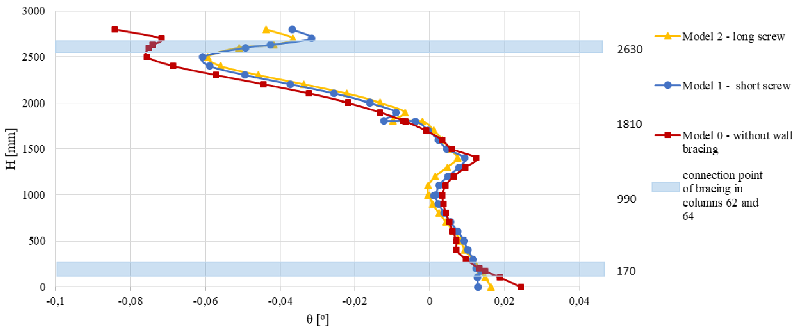

Fig. 8. The course of variability of the torsion angle along the length of the column shelf 63

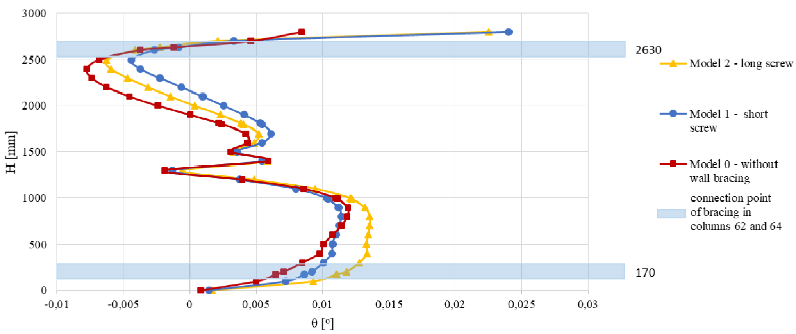

Fig. 9. The course of variability of the torsion angle along the length of the column web 64

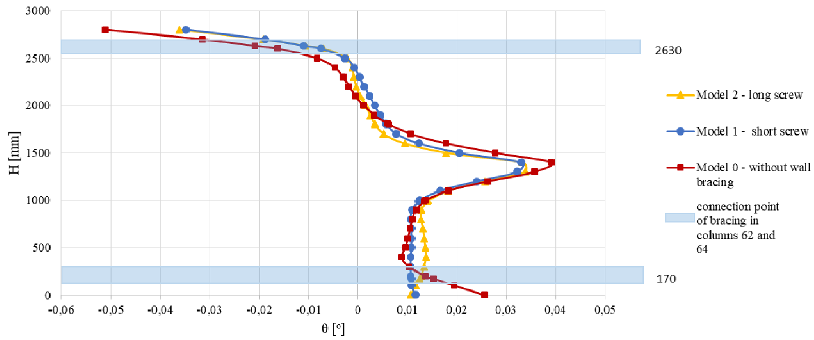

Fig. 10. The course of torsion angle variability along the length of the column shelf 64

4. Conclusions

Based on the results of the numerical analysis, it was shown that the building structure model without stiffening plates in the ground floor exceeded the horizontal values of horizontal displacements specified in the regulations.

The numerical analysis of the separated part of the wall makes it possible to assess the impact of the applied wall bracing on bending - torsional displacements. The results of the analysis of data were contained in the range of torsion angles by entering vertical bracing and the method of their attachment.

References

[1] Piechnik S., Pręty cienkościenne - otwarte: podręcznik dla studentów wyższych szkół technicznych, Wydawnictwo Politechniki Krakowskiej, Kraków 2000.

[2] Katalog ocynkowanych profili stalowych SundaySystem, PPUH AmTech Sp. z o.o.

[3] PN-EN 1991-1-1 Eurokod 1: Oddziaływania na konstrukcje. Część 1-1. Oddziaływania ogólne. Ciężar objętościowy, ciężar własny, obciążenia użytkowe w budynkach.

[4] PN-EN 1991-1-3 Eurokod 1: Oddziaływania na konstrukcje. Część 1-3. Oddziaływania ogólne. Obciążenie śniegiem.

[5] PN-EN 1991-1-4 Eurokod 1: Oddziaływania na konstrukcje. Część 1-4. Oddziaływania ogólne. Obciążenia wiatru.

[6] PN-EN 1993-1-1:2006, Eurokod 3: Projektowanie konstrukcji stalowych - Część 1-1. Reguły ogólne i reguły dla budynków.

[7] PN-EN 1990:2004, Eurokod 0, Podstawy projektowania konstrukcji.

[8] Major M., Major I., Kosiń M., Numerical static analysis of the curtain wall with light steel structure, Transactions of the VSB - Technical University of Ostrava 2017, Civil Engineering Series Vol. 17.

[9] Major M., Major I., Kalinowski J., Kosiń M., Analysis of a selected node of a truss made of cold-rolled sections based on the finite element method, Transactions of the VSB - Technical University of Ostrava 2018, Civil Engineering Series Vol. 18.

[10] Huei-Huang Lee, Finite Element Simulations with Ansys Workbench 13 Schroft Development Corporation 2011, ISBN: 978-1-58503-653-0.

[11] Ansys-Workbench v. 18.1 system documentation.

[12] Palacz P., Major M., Analiza porównawcza przemieszczeń ustroju prętowego z użyciem programów ADINA, Autodesk Robot oraz RFEM, Zeszyty Naukowe Politechniki Częstochowskiej 2018, seria Budownictwo 24, 262-266.

[13] Poński M., Reliability analysis of curved on plane hoist beam using coupling of Ansys APDL and Matlab, Zeszyty Naukowe Politechniki Częstochowskiej 2018, seria Budownictwo 24, 285-290.

Numeryczna analiza stateczności dwukondygnacyjnego budynku wykonanego w technologii lekkiego szkieletu stalowego

STRESZCZENIE:
Przedstawiono numeryczną analizę budynku jednorodzinnego wykonanego w technologii lekkiego szkieletu stalowego. Dla przyjętego budynku sporządzono modele numeryczne, który poddano analizie stateczności za pomocą programu Robot Structural. W analizowanej konstrukcji zestawiono możliwe do wystąpienia obciążenia zgodnie z obowiązującymi normami. Dodatkowo dokonano analizy giętno-skrętnej dla wyodrębnionej części przegrody zewnętrznej. Analiza ta została wykonana w programie Ansys 18.1.1 Academic Research. Na podstawie uzyskanych wyników oceniono wpływ rozpatrywanych modeli na stateczność bryły budynku, jak również kąta skręcania wybranych elementów przegrody pionowej.

SŁOWA KLUCZOWE:
lekki szkielet stalowy; numeryka; elementy cienkościenne