UDC 624.014 **DOI:10.31650/2707-3068-2024-28-46-54**

DESIGN OF INDUSTRIAL HALLS WITH A STEEL STRUCTURE DUE TO AN ACCIDENTIAL DESIGN SITUATION - THE IMPACT OF A VEHICLE ON A FRAME COLUMN

[1]**Wojnar A.**, PhD, Assistant Professor
awojnar@prz.edu.pl, ORCID: 0000-0002-0537-3864
[2]**Chernieva O**., DSc., PhD, Associate Professor
chernieva@odaba.org.ua, ORCID: 0000-0002-4807-6421
[1]**Śleczka L.**, DSc, PhD, Associate Professor
sleczka@prz.edu.pl, ORCID: 0000-0002-8979-7073
[1]*Rzeszow University of Technology*
[2]*Odesa State Academy of Civil Engineering and Architecture*

Abstract. The article presents the effect of vehicle impact on the behaviour of the steel structure of an industrial hall. A portal industrial hall with a span of 18.0 m, a length of 42.0 m and a column height of 7.0 m was considered. The spacing of the frames - the main transverse systems - was assumed equal to 6.0 m. The case in which a truck hits a frame column was considered. A scenario was assumed in which the impact occurs when the vehicle leaves the manoeuvring area located at the industrial hall. Therefore, according to EN 1991-1-7, the speed of the vehicle is 10 km/h. The weight of the vehicle was assumed to be 20000 kg. The direction of impact was assumed perpendicular to the side wall of the hall. The calculations were carried out in two stages. The first - optimization of cross sections of beams and columns in a permanent design situation. Static analysis was used. The second - determination of internal forces from impact in an accidental design situation. Dynamic analysis was used. Hall structural systems made of HEA, HEB and IPE sections were considered. The pinned and rigid connection of the column to the foundation was considered. The calculations were performed using Autodesk's Robot Structural Analysis 2024 computer software.

Keywords: industrial hall, stability of portal frame, vehicle impact, accidental design situation, FEM analysis.

1. Introduction. In a general, typical case, the procedure for designing a building structure can be divided into the following stages:

- acceptance of the location of the building and determination of climatic load zones: snow load and wind action;
- determination of the values of permanent loads;
- adoption of the use category of the object and determination of the value of variable loads;
- making combinatorics of loads;
- adoption of the appropriate type of analysis, making static calculations and determining the values of internal forces, deflections and displacements of elements and nodes of the structure;
- verification of the structure's limit states: ultimate limit state, serviceability limit state and general stability of the structure.

Verification of the ultimate limit state consists in checking the stability of the most stressed members and elements of the structure, as well as the load capacity of nodes and connections. Verification of serviceability limit state consists in checking that the values of permissible deflections of elements and displacements of nodes are not exceeded. The above analyses are carried out in a permanent design situation relating to the usual conditions of use of the structure.

In addition, the stability of the structure and the stability of the elements of the structure should be checked in design situations: transient, incidental and, if required, in a seismic design situation.

The paper presents an analysis of checking the stability of the structure in an accidental design situation relating to the exceptional load, which is the impact of a vehicle on a structural element - a column of the main load-bearing system. The calculations were carried out on the example of a single-bay industrial hall with a steel structure.

The dimensions of the hall are: width 18.00m, height at the ridge 8.00m, height at the eaves 7.00m, length 42.00m, spacing of the main support systems every 6.00m (Fig. 1).

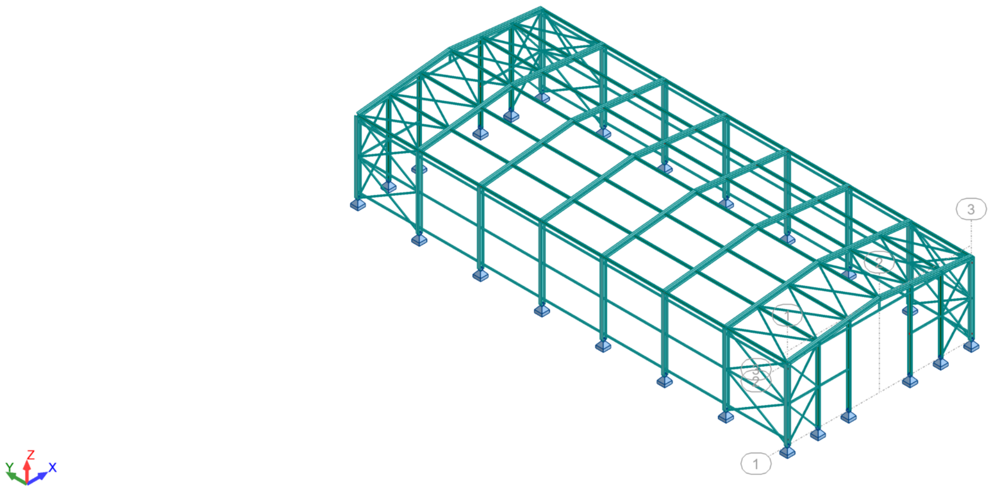


Fig. 1 Structure of the industrial hall - 3D view

2. Computational analysis

2.1. Loads

In the permanent design situation (ULS check), the following values of loads and actions (characteristic values) were assumed:

- permanent load: self-weight of the steel structure, weight of purlins and roof sheathing of 0.30kN/m^2;
- wind load - 1 wind load zone, base value of base wind speed $v_{b,0}$=22m/s, base speed pressure q_b=0.30kN/m^2;
- snow load - zone 2 ground snow load, s_k=0.9kN/m^2;

In the accidental design situation, the load from the impact of the vehicle on the column of the main frame structure was assumed. This is described in detail later in the paper.

Load combinations were adopted in accordance with the standards:

- EN 1990 Eurocode. Fundamentals of design;
- PN-EN 1991-1-1 Eurocode 1. Actions on structures. Part 1-1: General actions. Volumetric weight, dead weight, imposed loads on buildings.
- EN 1991-1-7 Eurocode 1 Actions on structures. Part 1-1: General actions. Exceptional actions.

2.2. Analysis

For the computational analysis, spatial models of the industrial hall's steel structure were created. Columns and beams were modelled using beam-type bar elements with 6 degrees of freedom at the node. The vertical and horizontal bracing rods were given attributes to allow them to carry only longitudinal forces (compression/tension).

Static calculations were performed using first-order linear analysis.

The computational analysis was carried out on a plane frame isolated from the model. Frames whose columns and beams are made of HEA, HEB and IPE profiles were considered. Pinned and rigid fixing of the column in the foundation was considered, this gave a total of 6 computational models (see Table 1).

The analysis was carried out in two steps:

- checking the stability of structural elements in a permanent design situation,
- checking the stability of structural elements in an accidental design situation.

Table 1. Types of computational models of the framework adopted for the analysis

Calculation model name	The method of connecting the column to the foundation	Column – type of profile	Beam – type of profile
Model 1 - Frame 1	pinned	HEA	HEA
Model 2 - Frame 2	rigid	HEA	HEA
Model 3 - Frame 3	pinned	HEB	HEB
Model 4 - Frame 4	rigid	HEB	HEB
Model 5 - Frame 5	pinned	IPE	IPE
Model 6 - Frame 6	rigid	IPE	IPE

2.2.1. Checking the stability of structural elements in a permanent design situation

In the first stage of calculations, the general stability of the steel frames of the hall structure was checked. Strength calculations were carried out assuming the following parameter values:

Frame column

- the critical length of the column for buckling in the plane of the frame was determined by linear buckling analysis as for sway frames, $L_{cr,y} > L_0$, where L_0 is the height of the column,
- the critical length of the column in the case of buckling out of the plane of the frame was determined as for non-sway frames; no exact calculation analysis was carried out, $L_{cr,z} = L_0$ was assumed,
- the length for calculating the lateral-torsional buckling was assumed equal to half the height of the column $L_{cr,LT}$ = 3.50m, it was assumed that the connection of the transom to the column is sufficiently rigid and protects the cross-section of the column from rotation at the point of connection with the wall housing transom.

Frame rafter

- the critical length of the rafter for buckling in the plane of the frame was determined on the basis of linear buckling analysis, as for sway frames, $L_{cr,y} > L_0$, where L_0 is the length of the rafter,
- the critical length of the rafter for buckling out of the frame plane was determined on the assumption that the connections of the rafter flange to the purlins limit the buckling length of the rafter; no detailed calculation analysis was performed, $L_{cr,z} = 0.33*L_0$ was assumed,
- the length for calculating the lateral-torsional buckling was assumed with the assumption that the connections of the transom flange with the purlins limit the lateral-torsional length of the transom; $L_{cr,LT} = 0.33*L_0$ was assumed.

Optimization of the structure's sections was carried out. As a criterion, the minimum weight of individual elements while maintaining their stability was used.

The results of the calculations are presented below, see Table 2.

Table 2. Results of calculations in the form of values of size and type of used steel profiles, frame weight and painting area

Calculation model name	Natural frequency	Column	Rafter	Frame weight [kg]	Painting area [m^2]
Frame 1	f=2,05Hz	HEA 340 Ratio: 0,95	HEA 320 Ratio: 0,88	3239	56,95
Frame 2	f=4,17Hz	HEA 300 Ratio: 0,92	HEA 300 Ratio: 0,91	2836	55,25
Frame 3	f=1,89Hz	HEB 320 Ratio: 0,88	HEB 280 Ratio: 0,97	3644	54,13
Frame 4	f=3,97Hz	HEB 280 Ratio: 0,84	HEB 280 Ratio: 0,83	3308	52,03
Frame 5	f=2,84Hz	IPE 500 Ratio: 0,89	IPE 450 Ratio: 0,88	2676	53,50
Frame 6	f=6,71Hz	IPE 500 Ratio: 0,78	IPE 450 Ratio: 0,80	2676	53,50

2.2.2. Checking the stability of structural elements in an accidental design situation

The analysis of the behavior of the structure due to vehicle impact (the second stage of calculations) was carried out based on the procedures described in PN-EN 1991-1-7:2006, Eurocode 1, Actions on structures, Part1-7: General actions, Exceptional actions. The value of dynamic force from impact, and the impact model were adopted according to Appendix C - Dynamic design for impact.

Impact is an interaction phenomenon between a moving object and a structure, in which the kinetic energy of the object is suddenly transformed into deformation energy. To find the forces of dynamic interaction, the mechanical properties of the object and structure are determined.

The engineering design of structures for impact usually uses equivalent static forces.

In advanced structural design, one or more aspects are considered:

- dynamic effects,
- nonlinear material behaviour.

Analysing Calculation procedure, the dynamics of the impact, a distinction is made between:

- hard impact, in which the energy is mainly dissipated by the striking body, the structure is designed as rigid,
- soft impact, in which the structure is designed as deformable and takes the energy of the impact.

In the calculations carried out, it was assumed that there is a hard impact. The dynamic effects of the impact were considered. The nonlinear behaviour of the material was neglected.

2.2.3. Dynamic design for impact - description of the calculation procedure

Assumptions:

- the structure is rigid and stationary,
- the impacting object deforms linearly during impact.

Calculation procedure:

1. The dynamic interaction force was calculated using the formula (C.1 EN 1991-1-7):

$$F_{DYN} = v_r\sqrt{k \cdot m}$$

v_r - velocity of the object at the moment of impact,
k - equivalent elastic stiffness of the object,
m - mass of the impacting object.

2. The force induced by the impact was taken as a rectangular impulse acting on the surface of the structure, see Figure 2. The duration of the pulse was calculated according to the formula (C.2 EN 1991-1-7):

$$\Delta t = \sqrt{\frac{m}{k}}$$

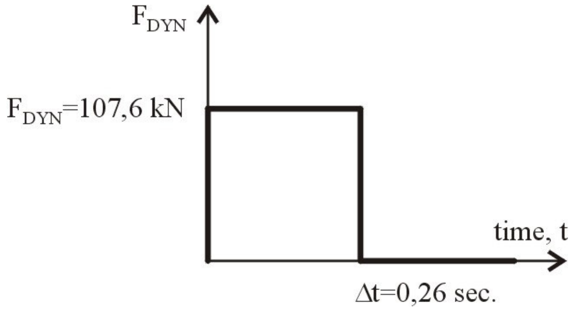


Fig. 2 Dynamic impulse from vehicle impact

3. The force caused by the impact of the vehicle is applied at a height of 0.50m for cars or from 0.50 - 1.50m for trucks.

The expression of pt. 1 gives the maximum value of the dynamic force on the outer surface of the structure. Inside the structure, these forces can lead to an increase in dynamic effects. An upper limit of these effects can be determined if it is assumed that the structure responds elastically and the load is realized as a step function, that is, a function that increases immediately to its final value and then remains constant.

Impact scenario.

- it is assumed that a 20000 kg heavy-duty delivery truck strikes a main support system frame column,
- the event takes place in the manoeuvring area located next to the hall,
- vehicle speed is assumed equal to 10 km/h = 2.78 m/sec,
- the direction of impact is perpendicular to the side wall of the hall,
- the magnitude of the dynamic force and the shape and duration of the pulse from the impact are shown below.

k = 300 kN/m - equivalent elastic stiffness of the object (vehicle stiffness, Table C.1 EN 1991-1-7),

m = 20000 kg - mass of the impacting object (truck, Table C.1 EN 1991-1-7).

$$F = v_r\sqrt{k \cdot m} = 2.78 \cdot \sqrt{300000 \cdot 20000} = 107.6\ kN$$

$$\Delta t = \sqrt{m/k} = \sqrt{20000/300000} = 0.26\ sec$$

2.3. Results of analysis (see Table 3,4)

Table 3. Values of internal forces and displacements calculated in Ultimate Limit State, Permanent Design Situation

Calculation model name	Column		Rafter		Horizontal displacement
	My [kNm]	N [kN]	My [kNm]	N [kN]	Ux [mm]
Frame 1	338,11	134,94	338,11	61,35	54
Frame 2	296,86	128,66	296,86	83,51	15
Frame 3	350,68	137,22	350,68	63,20	59
Frame 4	300,43	131,32	300,43	84,50	14
Frame 5	339,97	131,77	339,97	61,39	35
Frame 6	304,06	127,62	304,06	86,19	6

Table 4. Values of internal forces and displacements calculated in Accidental Design Situation

Calculation model name	Column		Rafter		Horizontal displacement
	My [kNm]	N [kN]	My [kNm]	N [kN]	Ux [mm]
Frame 1	110,19* 99,77**	18,96	110,85	14,72	74* 47**
Frame 2	101,76* 94,18**	3,63	23,70	3,70	5* 3**
Frame 3	110,31* 99,91**	20,50	109,62	15,05	78* 49**
Frame 4	103,40* 94,18**	4,72	31,50	4,17	5* 3**
Frame 5	108,41* 99,86**	11,77	93,99	13,23	46* 29**
Frame 6	99,90* 94,68**	2,15	7,39	2,71	2* 1**

* dynamic analysis, ** static analysis

The results of the computational analyses are presented in the form of:

- values of the bending moment My in the frame column and transom,
- values of longitudinal force N in the frame column and transom,
- the value of the horizontal displacement of the frame corner Ux,
- values of the natural frequency of the frame.

2.4. Obtained results

2.4.1 Ultimate Limit State, Permanent Design Situation

Considering the weight of the frame structure (Table 2, Fig. 3):

- The structure in which columns and beams are made of IPE profiles is the most favorable. Due to the step change in the height/size of IPE profiles, the method of fixing the column in the foundation did not affect the size of the profiles used (column IPE 500, beam IPE 450). However, the method of fixing the column in the foundation had an impact on the post-tensioning of individual sections - ratio= about 0.9 in case of pinned joint and ratio= about 0.80 in case rigid joint.

- The most disadvantageous is the structure in which columns and rafters are made of HEB profiles, with an articulated connection of the column to the foundation. In the case of a rigid connection of the column to the foundation, the mass of the frame made of HEB profiles is slightly lower, but still greater than the mass of frames made of HEA and IPE profiles.

Considering the painting area of the frame structure (Table 2, Fig. 4):

- The most advantageous is the construction in which columns and transoms are made of HEB profiles, with a rigid connection between the column and the foundation.

- The most unfavorable is the construction in which columns and transoms were made of HEA profiles, with an articulated connection between the column and the foundation.

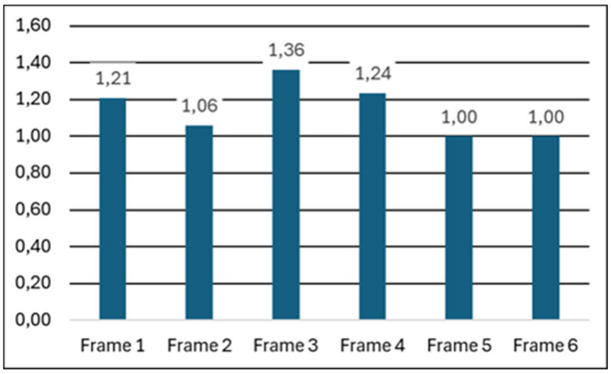


Fig. 3 Multiplicity of change of weight the frame in relation to the base frame - Frame 6 (Table 1) [number 1 is 2676 kg]

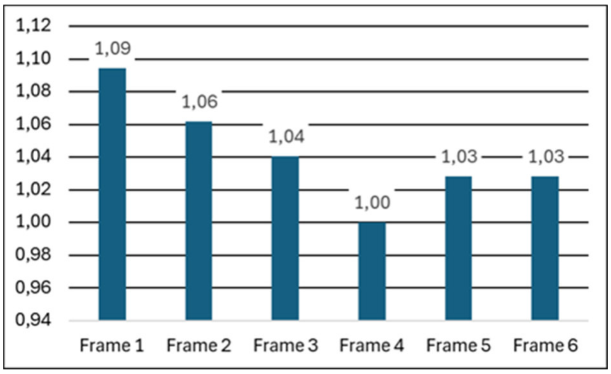


Fig. 4 Multiplicity of change the area of painting of the frame in relation to the base frame – Frame 4 (Table 1) [number 1 is 52,03 m^2]

Considering the horizontal displacement of the frame corner and the natural frequency (Table 2, 3):

- The highest stiffness is characterized by a structure in which columns and transoms are made of IPE profiles, with the column rigidly fixed in the foundation. This frame is characterized by the lowest value of horizontal displacement of the frame corner and the highest value of natural frequency.

- The least rigidity is characterized by the structure in which the columns and transoms were made of HEB profiles, with an articulated fixing of the column in the foundation. This frame is characterized by the highest value of the horizontal displacement of the frame corner and the lowest value of the natural frequency of vibration.

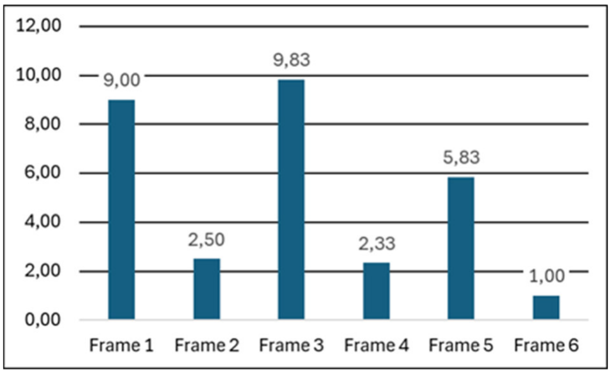


Fig. 5 Multiplicity of change of horizontal displacement of the frame corner in relation to the base frame - Frame 6 (Table 1)
[number 1 is 6 mm]

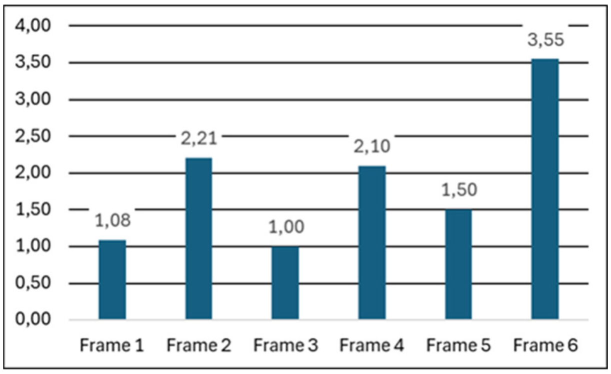


Fig. 6 Multiplicity of change of natural frequency of the frame in relation to the base frame - Frame 3 (Table 1)
[number 1 is 1,89 Hz]

2.4.2 Accidental Design Situation

Analysing the values of internal forces in the frame structure generated from the impact of the vehicle and comparing them with the values of forces at the ultimate limit state, it was found:

- Longitudinal forces reach relatively small values and can be ignored in future analyses.
- For the considered impact scenario, the value of the bending moment in the column caused by the impact of the vehicle reaches about 1/3 of the value of the bending moment calculated at the ultimate limit state and does not pose a significant threat to the safety of the structure.
- For the considered impact scenario, the values of the bending moment in the frame transom caused by the impact of the vehicle on the frame column, changes significantly depending on how the column is fixed in the foundation:
- in the case of an articulated joint, represents about 1/3 of the bending moment calculated at the ultimate limit state and does not pose a significant threat to the safety of the structure,
- in the case of a rigid connection is negligibly small.

Table 4 also compares the values of the bending moment in the column and the horizontal displacement of the frame corner, from the impact of the vehicle on the frame column, calculated using static and dynamic analysis. The values obtained by dynamic analysis are higher by about 10% for the bending moment and up to about 50% for the horizontal displacement of the frame corner. It should be noted, however, that this difference will increase as the vehicle speed increases, accompanied by an increase in the dynamic force from impact.

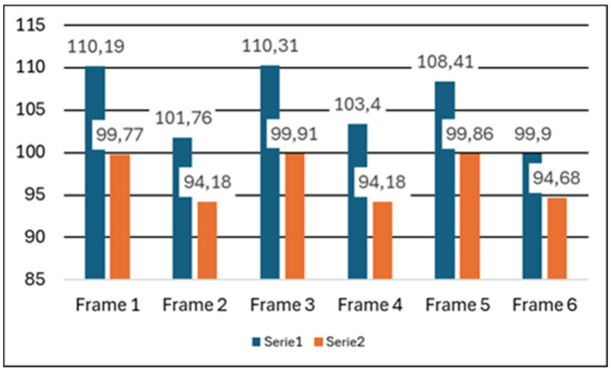


Fig. 7 Values of the bending moment in the column of the frame calculated using dynamic analysis (Serie 1) and static analysis (Serie 2)

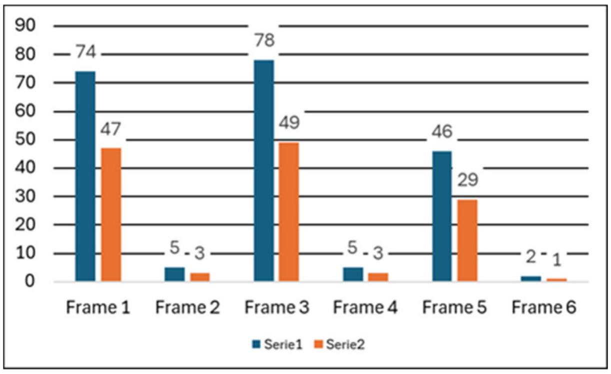


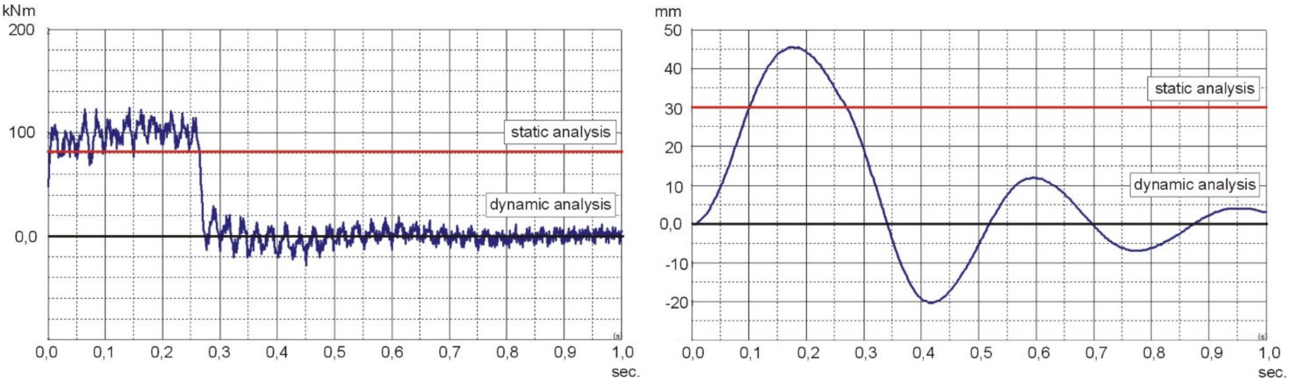
Fig. 8 Values of the horizontal displacement of the frame corner calculated using dynamic analysis (Serie 1) and static analysis (Serie 2)

Conclisions

In conclusion, it can be said that at low vehicle speeds the impact is not dominant due to the stability of the structure. In the case of frames made of HEA, HEB and IPE profiles, the internal forces in the structure and their distribution in individual bars did not pose a serious threat to the safety of the hall.

The final part of the paper also compares the values of the bending moment in the column and the horizontal displacement of the frame corner caused by the impact, obtained using static and dynamic analysis. It can be noted that static analysis does not take into account dynamic effects, resulting in the determination of underestimated values of forces and displacements. These differences increase as the mass and speed of the striking object increases.

In order to use static analysis effectively, it is required to assume equivalent forces from impact in the calculations - which is described in more detail in EN 1991-1-7.



Bending moment diagram — Horizontal displacement diagram

Fig. 9 Comparison of bending moment values in the column and horizontal displacement of the frame corner determined using static and dynamic analysis (same value of impulse from vehicle impact) - example diagrams.

References

[1] EN 1990. Eurocode 0 - Basis of structural design. European Committee for Standardisation, (2002).
[2] EN 1991-1-1: Eurocode 1: Actions on structures - Part 1-1: General actions -Densities, self-weight, imposed loads for buildings. European Committee for Standardisation, (2002).
[3] EN 1991-1-3: Eurocode 1: Actions on structures – Part 1-3: General actions – Snow loads. European Committee for Standardisation, (2003).
[4] EN 1991-1-4: Eurocode 1: Actions on structures – Part 1-4: General actions - Wind actions. European Committee for Standardisation, (2005).
[5] EN 1991-1-7: Eurocode 1 - Actions on structures - Part 1-7: General actions - Accidental actions. European Committee for Standardisation, (2006).
[6] EN 1993-1-1: Eurocode 3 - Design of steel structures - Part 1-1: General rules and rules for buildings. European Committee for Standardisation, (2006).
[7] Autodesk Robot Structural Analysis Professional ver. 2024

ПРОЕКТУВАННЯ СТАЛЕВОГО КАРКАСУ ПРОМИСЛОВОГО ЦЕХУ НА ВИНИКНЕННЯ АВАРІЙНОЇ РОЗРАХУНКОВОЇ СИТУАЦІЇ - ЗІТКНЕННЯ ТРАНСПОРТНОГО ЗАСОБУ З КОЛОНОЮ КАРКАСУ

[1]**Вожнар А.,** к.т.н.,
awojnar@prz.edu.pl, ORCID: 0000-0002-0537-3864
[2]**Чернєва О.,** к.т.н., доц.
chernieva@ogasa.org.ua, ORCID: 0000-0002-4807-6421
[1]**Сленчка Л.**, к.т.н., доц.,
sleczka@prz.edu.pl, ORCID: 0000-0002-8979-7073
[1]*Жешувський технологічний університет*
[2]*Одеська державна академія будівництва та архітектури*

Анотація. У статті представлено результати впливу автотранспорту на поведінку сталевої конструкції промислового цеху при зіткненні. Розглядався портальний промисловий цех з прольотом 18,0 м, довжиною 42,0 м і висотою колони 7,0 м. Відстань рам - основних поперечних систем - прийняли рівним 6,0 м. Розглядався випадок наїзду вантажівки на колону каркасу. Був припущений сценарій, в якому зіткнення відбувається, коли транспортний засіб залишає зону маневрування, розташовану в промисловому цеху. Отже, згідно з EN 1991-1-7, швидкість транспортного засобу становить 10 км/год. Вага автомобіля передбачалася 20000 кг. Напрямок удару вважався перпендикулярним бічній стіні цеха. Розрахунки проводились у два етапи. Перший етап виконувався за допомогою статичного аналізу та передбачав оптимізацію перерізів балок і колон в постійному проектному положенні. Другий етап стосувався визначення внутрішніх сил від удару в аварійній розрахунковій ситуації та проводився з використанням динамічного аналізу. В процесі дослідження були розглянуті конструктивні системи з поперечним перерізом, складеним з нормальних двотаврів із паралельними гранями полиць (IPE) та з двотаврів з широкими полицями (HEA, HEB). Враховані варіанти шарнірного та жорсткого з'єднання колони з фундаментом. Розрахунки проводилися за допомогою комп'ютерної програми Autodesk Robot Structural Analysis 2024. Результати дослідження показали, що на низьких швидкостях автомобіля вплив не є домінуючим завдяки стійкості конструкції. У випадку каркасів з профілів HEA, HEB та IPE внутрішні сили в конструкції та їх розподіл в окремих стрижнях не становили серйозної загрози для безпеки цеху.

У статті порівнюються значення згинального моменту в колоні та горизонтального зміщення кута рами, викликаного ударом, отримані за допомогою статичного та динамічного аналізу. Можна відзначити, що статичний аналіз не враховує динамічні впливи, в результаті чого визначаються занижені значення сил і переміщень. Ці відмінності зростають із збільшенням маси та швидкості об'єкта, що здійснює удар.

Для ефективного використання статичного аналізу необхідно припустити еквівалентну силу удару в розрахунках, що описано більш детально в EN 1991-1-7.

Ключові слова: промисловий цех, стійкість портальної рами, удар транспортного засобу, аварійна розрахункова ситуація, аналіз методом кінцевих елементів.