Technical Note
SELECTION OF FINITE ELEMENT FOR DESIGNING METAL BUILDING STRUCTURES

D. O. Kuzmenko
Master of Engineering Science (Belarusian-Russian university, the Republic of Belarus)

V. M. Fridkin
Doctor of Engineering Science, professor (Moscow State University of Railway Transport of Emperor Nicholas II)

I. M. Kuzmenko
Candidate of Science, assistant professor (Belarusian-Russian university, the Republic of Belarus)

INTRODUCTION

Modern engineering buildings have various types and design variants (Bogdanov 2011, 6–16; Kuzmenko 2003, 47–50; Kuzmenko 2006, 198–202; Kuzmenko 2014; Kuzmenko, Fridkin, & Markov 2015, 40–51; Kuzmenko & Fridkin 2017, 171; Fridkin 2011, 171; Fridkin, Chesnokov, & Kokosadze 2015, 119). Load-carrying elements of the metal engineering structures are designed as “linear” elements that are manufactured from metal sheets which are connected in tri-dimensional beams by “nodal” load-carrying elements (gusset plates). Each gusset plate connects adjacent beam elements as tri-dimensional beams and finally in tri-dimensional structures. Other than simple building structures (e.g. industrial building door frames that consist of door bolts, columns, trusses etc.) under model engineering which do not require high processing capacity, estimation of wide-span bridges or floor structures of entertainment facilities demands a vast amount of calculations.

Structures, representing linear systems, can be calculated with the help of volume, shell or beam finite elements. One can deal with complicated structures (that do not have a uniform section, but with structural elements with a complicated form, made from different materials) only with the help of volume finite elements, with some minor exceptions which can be analysed using shell elements. Nowadays there are several software programs that allow, within a certain period of time, calculation of any engineering structure for external loading.

The software programs of CAE/CAD systems can solve any problem within a short period of time; therewith they allow making changes to the structure if this is necessary after the calculations. The results of experimental verification of the accuracy of numerical models are presented in Lagerev (2014, 36-40). In Shimanovsky (2008, 61) the influence of the finite element size on the accuracy of statistical estimation was considered. The requirements that are specified in developing the computer models to the mesh definition of the finite elements are described in Bogdanov (2013, 119). However, there are no direct comparisons of the various types of finite elements and their influence on the accuracy, calculating speed and resource-intensiveness of the problem being solved in these publications.

THE MAIN TYPES OF THE FINITE ELEMENTS AND THEIR FEATURES

Finite element types used in various software programs, for example, in CAE / CAD Ansys Work Bench, are described in Basov (2002, 2005). From all the variety of types of finite elements (FE) for solving the static problem of calculating skeletal systems, in the opinion of the authors, it is sufficient to use FE types of BEAM, SHELL and SOLID. The size of the finite element of the BEAM, SHELL and SOLID types is standardized by the segment length between the nodes of the element. By that, it is meant that the whole element has a uniform section, geometrical characteristics and material characteristics.

All volume BEAM types have two nodes that can have three or six degrees of freedom on the main coordinate axes. All volume SHELL types have from four to eight nodes that can have three or six degrees of freedom on the main coordinate axes. The element SOLID type can change its shape (cube, tetrahedron, hexahedron), depending on the problem to be solved. All volume SOLID types have from eight to twenty nodes that can have three or six degrees of freedom on the main coordinate axes.

Comparative analysis of the accuracy and resource capacity of the solution of the problem of load-bearing linear elements of metal building structures can be done using finite elements BEAM 188, SHELL 181 and SOLID 186. We give a detailed description of each of these FE.

BEAM 188 (Figure 1a) – three-dimensional linear beam element with finite deformations. Element is suitable for direct modelling of beam structures with a moderate ratio of length to thickness. It is built on the basis of Timoshenko's beam. It takes into account the effects of tangential (shear) deformations. It has six or seven degrees of freedom in each node. This includes movements in the direction of the X, Y and Z axes and rotations around the X, Y and Z axes. Under certain conditions, a seventh degree of freedom (cross-sectional distortion) is added. This element is suitable for linear as well as nonlinear problems with large rotations and (or) large deformations.

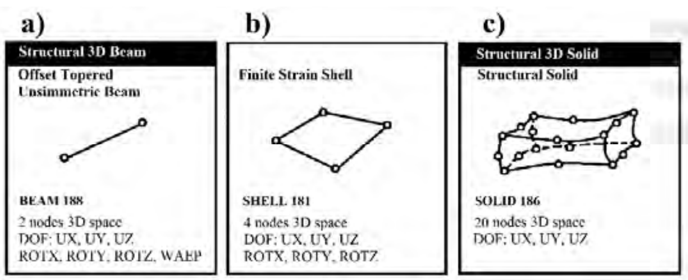

Figure 1. The finite elements used in the study.

The element has, by default, the ability to account for the change in stiffness when loaded. This makes it possible to use this element to study stability problems in compression, bending, and torsion (by applying eigenvalues and by studying the loss of stability by the method of searching along the length of the arc).

SHELL 181 (Figure 1b) – multilayer shell with finite deformations. The element is well suited for calculating shell models with small or moderate thickness. The element has four nodes and six degrees of freedom in each node: moves in the direction of the X, Y, and Z axes of the nodal coordinate system and rotations around the X, Y, and Z axes of the nodal coordinate system. Elements of triangular shape can be used only as transition elements in grids.

This element can be used in linear problems and in nonlinear problems with large rotations and (or) deformations. In nonlinear problems, the change in shell thickness is taken into account. As applied to the element, full and truncated variants of numerical integration are supported. The element can be used to calculate multilayer or three-layer shells.

SOLID 186 (Figure 1c) – three-dimensional (3D) element of a volume stress-strain state with twenty nodes. The element has a quadratic representation of the displacements and is able to use an irregular grid shape (for example, based on models imported from various CAD complexes). The nodes have three degrees of freedom each, moving in the directions of the X, Y, and Z axes of the nodal coordinate system. It can have an arbitrary orientation in space, has the properties of plasticity, hyper-elasticity, creep, changes in rigidity when applying loads, large displacements and large deformations.

It is possible to have a mixed formulation for the calculation of almost incompressible elastoplastic materials and completely incompressible hyper-elastic materials. To control the output of data, there are special options. In addition, these elements allow analysis of concentration of stresses.

When performing the calculations of the majority of the supporting bar elements, in our opinion, it is necessary to use the FE of type BEAM. When calculating the load-bearing bar elements with a thin wall, it is necessary to use the FE type BEAM or SHELL. The most accurate values for the calculation of structures with load-bearing bar elements of any complexity can be obtained using SOLID-type finite elements.

STATEMENT OF PROBLEM AND OBJECTS OF THE STUDY

In this paper, the possibility of using more "simple" types of finite elements (FE) is justified, without loss of accuracy of calculation. In this case, the calculation procedure will require less computing resources. This will make it possible to use fairly accurate and least resource-intensive solutions in future studies of large-span structures.

The test objects were considered the simplest bar bearing element as well as conventionally used Ishaped cross section (Figure 2, a) in metallic building structures. The selected dimensions are: h = 200 mm; b_f = 100 mm; t_f = 8 mm; t_s = 6 mm; and ℓ = 3000 mm. Material of the bar is low-alloy steel.

Numerical experiments were performed on solid, beam and shell models. A static analysis (Static Structural) and a calculation for loss of stability by its own value (Eigenvalue Buckling) were performed.

The results of the numerical experiment are compared with the values obtained by engineering calculation in Construction Regulation Standards Building Code (CRSBC) and Technical Code of Common Practice EN (TCP EN) (CRSBC II-23-81, 1990, 96; CRSBC 2.01-07-85, 1986, 36).

The boundary conditions were taken as follows: for a beam working on transverse bending, in accordance with Figure 2b; to calculate the bar for longitudinal bending (calculation for stability) – in accordance with Figure 2c. This approach made it possible to assess the advantages of using certain types of FE depending on the loading of the bar.

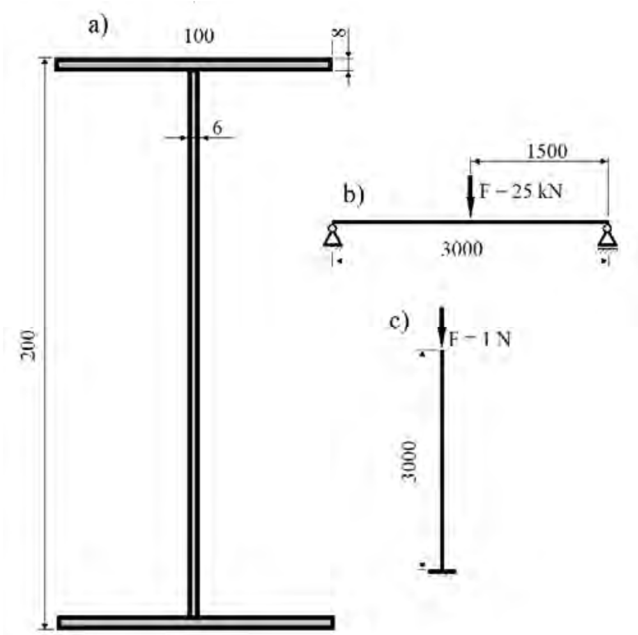

Figure 2. (a) Objects of research: cross-section of investigated models; (b, c) setting boundary conditions.

The sizes of the final elements are chosen as follows: for BEAM – 1 mm; for SHELL – 6 mm; for SOLID – 2 mm. In the study of the loss of stability, the dimensions of the final element were assumed to be equal: for shell FE (SHELL) – 4 mm and for volumetric FE (SOLID) 6 mm.

When using the SOLID type of FE, the normal stresses and deflections were determined in the lower (stretched) layer in the section where the force is applied. As parameters determining the load-carrying capacity of the bar in transverse bending, normal stresses and deflections are chosen. In the longitudinal bending of the bar (central compression), the critical forces and the values of the stability factors were determined.

THE ENGINEERING CALCULATIONS USING THE BUILDING CODE II-23-81*[14]

Strength Calculations: These calculations are based on the Construction Regulation Standards Building Code II-23-81*(CRSBC II-23-81, 1990, 96). According to the standard, when evaluating strength, the following condition must be met:

$$\frac{M}{W_{n,min}} \le R_y \cdot \gamma_c,$$

where M is the calculated value of the bending moment corresponding to the material reaching the yield point at a critical section of the bar;

$W_{n,\,min}$ is the moment of resistance of the section (see Figure 2a). When this cross-section is used in the elastic stage, we take $W_{n,\,min} = W_{pl} = 1.78689 \cdot 10^{-4}$ m^3;

R_y is the yield strength for steel (we take this as 250 MPa);

γ_c is coefficient of conditions of the design. It is assumed to be 1.1 (see Table 6 [14]).

The values of the calculated and acting moments in the middle of the span (see figure 2b) and the maximum normal stresses are shown in Table 1.

Calculation of Deflections: To ensure the deflections do not exceed the requirements, the following conditions must be met:

$$f \le f_y,$$

where f is the calculated maximum deflection of the beam; and f_y is the standard deflection of the beam. The ultimate deflection according to CRSBC 2.01.07-85 * (1986, 36) for the loading scheme (see Figure 2b) should not exceed L / 150. The Young's modulus for steel is assumed 206,000 MPa.

The results of calculating the bearing capacity for deflections are also presented in Table 1.

CALCULATION USING EUROCODE EN 1993-1-1-2009, P. 6.2.5 (EN 1993-1-1-2009, 2010, 85):

Strength calculations. The design value of the bending moment M_{Ed} in each cross-section must satisfy the condition:

$$\frac{M_{Ed}}{M_{c,Rd}} \leq 1{,}0,$$

where $M_{c,\,Rd}$ is the effective value of the bending moment. It is based on the plastic stage of operation $M_{pl,\,Rd}$, and also the partial safety factor γ_{Mo}. In accordance with 6.1 (1) B and Table NP.4 [15], γ_{Mo} = 1.025 / 1.1 = 0.932.

By calculation, the specified values of the moments and normal stresses correspond to the values obtained by the method of CRSBC II-23-81 * (see Table 1).

Deflection limit state. The ultimate deflection in this case should not exceed L / 300. Load deflection (adopted earlier in the CRSBC) f = δ = 0.00382 m. Limit deflection according to the norm δ_{limit} = 3/300 = 0.01 m. Thus, the deflection from the action of force does not exceed the normative one.

Calculation of the bar for loss of stability.

The calculation was carried out using traditional methods of material mechanics. Critical Strength (F_{cr}) is a load exceeding which causes the loss of stability of the original form (position) of the body. From the moment of the onset of the critical state to the moment of failure, the systems develops deflections extremely rapidly. In this way, when calculated for stability, the critical load is similar to the breaking load when calculated for strength. The stability condition is written in the following form:

$$F_{max} \leq F_{cr.}$$

The flexibility of the I-beam cross-section (see Figures 2a,c) λ = 270.3, and the ultimate flexibility of steel λ_u = 100.8. For $\lambda > \lambda_u$, the critical force was determined by Euler's formula: F_{cr} = 75.35 kN.

Table 1. Results of the calculation of the bar for transverse bending according to the methods of CRSBC II-23-81 * (clause 1) and TCP EN 1993-1-1-2009 (clause 2).

№	The design moment **M**, [N·m]	Actual moment M_d, [N·m]	Maximum normal stress σ, [MPa]	Normative deflection f_y, [m]	Limiting deflection **f**, [m]
1	49139.48	18750	104.93	0.00382	0.02
2	54822.96	18750	104.93	0.00382	0.01

CALCULATION OF THE BEAM USING THE ANSYS WORK BENCH PROGRAM

Figure 3 shows the finite element mesh partitioning and boundary conditions for the solution of the problem, using 3D (finite element type SOLID).

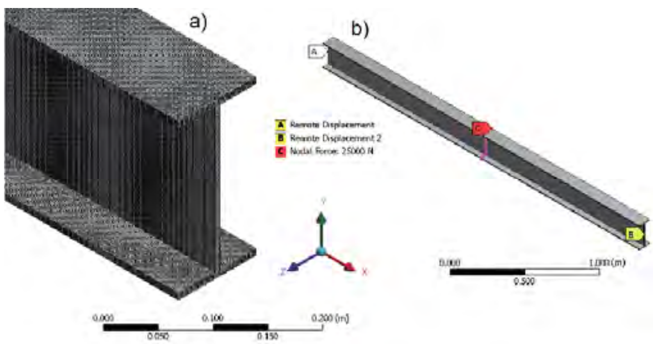

Figure 3. To the calculation of the bar for bending: the fragment of the partitioning of the I-beam into finite elements (a); boundary conditions (b).

For models in which other types of finite elements were used, the boundary conditions were set similarly. The initial load (see Figure 3b, Marker C) was set with the condition that the resulting stresses do not exceed the tensile-compressive yield strength of steel, which is assumed to be 250 MPa for the basic calculation in ANSYS WB. The ultimate tensile strength is 460 MPa.

Support of the bar (see Figure 3b, Markers A and B) was made at the point of the geometric centre of gravity of the cross-section for all models, while for the model with a finite element of the SHELL type three sections of the section (corresponding to two flanges and the I-beam web) were fixed; for FE type

SOLID, the fastening was made behind the end of the beam on the surface, and for the FE type of BEAM, at the extreme points of the element. The results of the calculations are presented in Table 2 and in Figure 4.

Table 2. Results of the numerical study of the work of the bar on bending with the use of FE of different types.

	Finite element type			**Engineering calculation**
	BEAM 188	**SHELL 181**	**SOLID 186**	
Deflection, [mm]	4.1506	4.2209	4.1605	3.82
Normal stresses, [MPa]	104.93	102.40	104.48	104.93
Δ in deflection, %	7.98	10.49	8.91	
Δ on the basis of stresses, %	0	2.41	0.43	

Note: the Δ symbol in Tables 2 and 3 indicates the percentage discrepancies between the results of FEM calculations and engineering calculations for CRSBC and TCP EN.

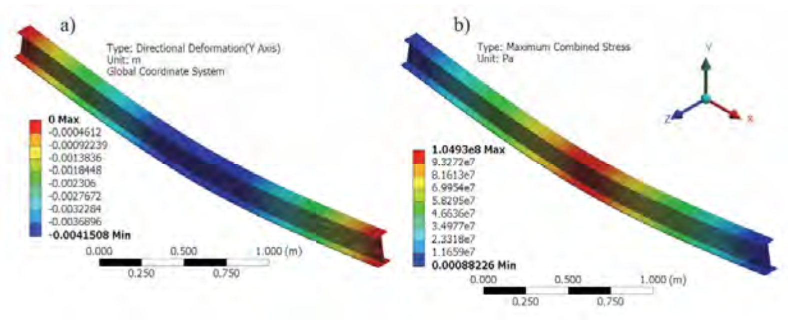

Figure 4. Results of the calculation of the bar for bending (FE of the BEAM type): deflection a); normal stresses b).

BAR FOR LOSS OF SUSTAINABILITY IN THE SOFTWARE COMPLETE ANSYS WB

We consider the finite element mesh partitioning and boundary conditions of the problem, using the volumetric (SOLID) type of finite elements (Figure 5). The support of the bar (see Figure 5b, Marker A) was set at the point of the geometric centre of gravity of the cross section for all models. In this case, for the model with the FE-type SHELL, three faces of the section were fixed (corresponding to the two flanges and the web), for the SOLID type of model, the boundary was set along the surface of the support section, and for FE of BEAM type, at the end point of the element. The results of the calculations are given in Table 3.

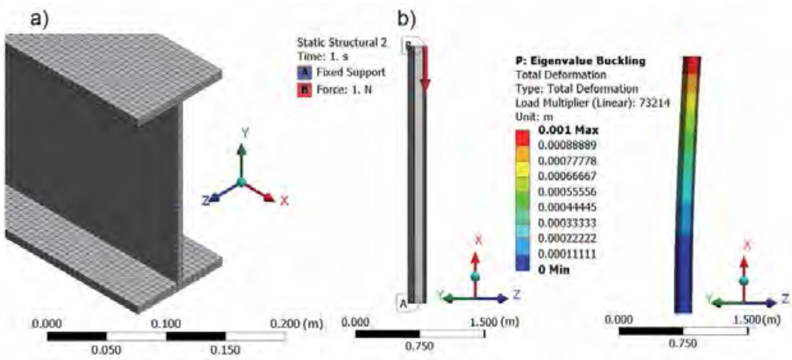

Figure 5. The results for the bar for longitudinal bending, (type of SOLID): a fragment of the mesh of the I-beam into finite elements (a); boundary conditions (b); and loss of stability (c).

Table 3. Results of a numerical study of the operation of the beam on the loss of stability with the use of FE of different types.

	Finite element type			Engineering calculation
	BEAM 188	SHELL 181	SOLID 186	
Critical force, [N]	73312	73200	73214	75350.7
∆ by the critical force, %	2.71	2.85	2.84	

RESOURCE REQUIRED FOR THE CALCULATIONS DEPENDING ON THE TYPE OF FE

The resource intensity of the calculation is determined by the time required to complete the calculation, the amount of information the computer operates (the amount of computational RAM, the disk space for calculating and storing equations, the size of the file with the results of the research), and also by the productivity (the computational frequency of the solver of the equations). The calculations were performed on a computer with the following characteristics:

- Intel (R) Core (TM) processor i7-4790K CPU @ 4.00GHz;
- Ansys version 17.1
- Calculation modes: a) Static Structural; B) Eigenvalue Buckling

Table 4 shows the results of a comparative analysis of the computational parameters required to calculate the beam for bending.

Table 4. The resource intensity of the calculation, depending on the type of the final element.

Analysis parameter	Finite element type		
	BEAM 188	**SHELL 181**	**SOLID 186**
Number of cells FE	3000	18424	328000
Number of nodes FE	6001	18852	1718506
The required amount of RAM, [MB]	2112	2112	6421
Required disk space for calculation, [MB]	57	329	6420.99
The size of the results file, [MB]	7,9375	21.75	490.63
The computing frequency of the equation solver, [MFlops]	1670	18480	13034.4
Total processor time, [s]	2	2.8	451

As can be seen from the table, the use of finite elements of the BEAM type has more advantages, with respect to other types (in this case we will make a reservation, with one size considerably exceeding others), both in comparison with engineering calculation and in comparison with saving the PC resource.

CONCLUSION

1. The use of modern software systems based on CAE / CAD systems, and sufficiently powerful personal computers, allows optimizing the calculation of building structures.

2. Comparison of calculation results in the Ansys WB software package and engineering calculation (Table 2) that the closest values of the normal stresses and deflections of the I-beam were obtained using finite elements such as BEAM and SOLID. The finite SHELL element has a greater percentage of discrepancy, and is not recommended for use.

3. When studying the operation of the bar for stability, it is possible to use the FE of all the types considered, although there is some advantage in the FE type of BEAM (Table 3).

4. The use of FE type BEAM significantly facilitates the work of a personal computer, without loss of accuracy of the calculation itself, especially when calculating large-span structures with a large number of elements of constant cross-section, without requiring large computational resources (Table 4).

5. When calculating the FEM based on CAE / CAD systems of bar elements, one of the dimensions of which considerably exceeds the cross-sectional dimensions, the optimal type of FE is BEAM.

REFERENCES

Basov, K. A**.** (2002). *ANSYS in examples and problems*. Moscow: Computer-Press (In Russian).

Basov, K. A. (2005). *ANSYS: user's guide*. Moscow: DMK Press (In Russian).

Bogdanov, S. V**.** (2011). Features of the design of fast-erecting welded walls of vertical cylindrical tanks of composite bearing elements. Mogilev: *Vestn. Belarusian-Russian University*, 3, 6-16. (In Russian).

Bogdanov, S. V. (2013). Experimental verification of the accuracy of numerical models in the study of deformation fields in fast-erecting objects from composite bearing elements. In Kuzmenko, I. M., Kuzmenko, D. O. & Cikunova, T. V. (eds.) *New materials, equipment and technologies in industry: materials of the Intern. Scientific-techn. Conf. Young scientists*. Mogilev: Belarusian-Russian Univ., 119. (In Russian).

CRSBC II-23-81 (1990). Steel structures / Gosstroy USSR. M.: TsITP Gosstroy USSR (In Russian).

CRSBC 2.01-07-85 (1986). Loads and effects / Gosstroy USSR. M.: TsITP State Building of the USSR (In Russian).

EN 1993-1-1-2009 (2010). Eurocode 3. Design of Steel Constructions. Part 1-1. General rules and rules for buildings: TCP (In Russian).

Fridkin, V. M**.** (2011). *Formation of building structures*. Moscow: MGSU (In Russian).

Fridkin, V. M**.,** Chesnokov, S. A. & Kokosadze, A. E. (2015). *Engineering facilities of underground power engineering*. M.: ASV (In Russian).

Kuzmenko, I. M. (2003). Application of welded load-bearing elements in new composite building structures. In Kuzmenko, I. M., Pavlyuk, S. K. & Fridkin, V. M. (eds.) *Welding* 9, 47–50. (In Russian).

Kuzmenko, I. M. (2006). Aspects of designing a composite bearing element using CAD. In Fridkin, V. M., Podimako, M. E., & Leonenko, O. V., Medvedev, V. N. (eds.) Mogilev: *Vestn. Belarusian-Russian University* 4, 198-202. (In Russian).

Kuzmenko, I. M. (2014). Experience of application in the Republic of Belarus of new constructive solutions of span structures of road bridges. In Fridkin, V. M., Sisa, N. S. & Markov, S. N. Naukovedenie 5(24). Retrieved 18 Jan 2018 from https://naukovedenie.ru/PDF/21KO514.pdf. (In Russian).

Kuzmenko, I. M., Fridkin, V.M., & Markov, S. N. (2015). Belarusian-Russian innovation: creation of engineering structures of the XXI century. *Proceedings of the METNET Seminar 2015 in Budapest. Häme University of Applied Sciences*, 40-51.

Kuzmenko, I. M. & Fridkin, V. M. (2017). *Prospects for the development of building structures of engineering structures*: monograph. Mogilev: Belarusian-Russian Univ. (In Russian).

Lagerev, I. A. (2014). Influence of the size of the finite element on the accuracy of the static calculation of the load-bearing metal structure of the overhead crane. *Journal of Science and Modernity* 32(2), 36–40 (In Russian).

Shimanovsky, A. O. (2008). Application of the finite element method in solving problems of applied mechanics: the teaching method. In Shimanovsky, A. O. & Putyato, A. V. (eds.) *Manual for students of technical specialties*. Ministry of Education of the Republic of. Belarus, Byelorussia. State. Un-t trans. Gomel: BelGUT. (In Russian).