MINISTRY OF EDUCATION AND SCIENCE OF UKRAINE

O. M. BEKETOV NATIONAL UNIVERSITY
of URBAN ECONOMY in KHARKIV

P. Reznik, O. Lugchenko

COMPUTER METHODS FOR CALCULATING BUILDING STRUCTURES

LECTURE NOTES

(for second (master's) level of higher education, full-time and part-time study in specialty 192 – Building Industry and Civil Engineering, educational program "Industrial and Civil Engineering")

Kharkiv
O. M. Beketov NUUE
2025

УДК 624.01:004.942

Reznik P. Computer methods for calculating building structures: Lecture notes for students of the second (master’s) level of higher education full-time and part-time forms of study in specialty 192 – Building Industry and Civil Engineering, educational program “Industrial and Civil Engineering”/ P. Reznik, O. Lugchenko ; O. M. Beketov National University of Urban Economy in Kharkiv. – Kharkiv : O. M. Beketov NUUE, 2025. – 109 p.

Authors:

PhD (Tech.), Associate Professor P. Reznik,

PhD (Tech.), Associate Professor O. Lugchenko

Reviewer

O. Kalmykov, Ph.D in Technical Sciences, Associate Professor of the Department of Structural Engineering O. M. Beketov National University of Urban Economy in Kharkiv

Recommended by the Department of Building Structures, Meeting Minutes, record 8 on January 16, 2025

CONTENTS

INTRODUCTION

The modern construction complex is characterized by the wide use of computer modelling tools, which make it possible to process complex structures at the stage of design and analysis. Software packages built on the Finite Element Method (FEM) provide a high level of detailing and the possibility of considering nonlinear, dynamic, and Multiphysics problems. They support regulatory requirements (Eurocode), contain material libraries and typical load models, and can also be integrated with the BIM environment.

Based on the results of studying the discipline, students must be able to idealize and model building structures, assign boundary conditions and loads, perform linear and nonlinear analyses, examine dynamic behaviour, and apply modern software packages (LIRA-FEM, LIRA-10, RFEM 6, SCAD, ANSYS, Abaqus, etc.) for the design of building elements. The training is intended to develop in the future engineering the ability to critically analyse calculation results and make decisions regarding the optimization of structural solutions.

1.1 Purpose, objectives, and significance of the discipline

The purpose of teaching the discipline “Modern Computer Methods of Structural Analysis” is to provide students with knowledge and skills in applying modern information technologies for the modelling, analysis, and design of building structures of varying degrees of complexity, using leading software packages such as LIRA-FEM, LIRA 10, RFEM 6, ANSYS, ABAQUS, and others. Special emphasis is placed on mastering the Finite Element Method (FEM) as a universal tool of engineering analysis in both linear and nonlinear formulations of structural mechanics problems.

The discipline contributes to a deeper understanding of the principles of computer modelling of structures, the rational selection of model types, the idealization of materials and structural elements, the setting of boundary conditions, the analysis of the results obtained, and the formation of practical skills in working with modern building calculation programs.

Main objectives of the discipline:

1. Introduction to modern construction design methodology and the role of the structural engineer in the digital construction process.
2. Studying the basics of the finite element method (FEM) in the context of practical application for structural calculations.
3. Forming an understanding of linear and nonlinear problems in structural theory, including physical, geometric, constructive, and genetic nonlinearity.
4. Mastering the principles of constructing calculation models, considering the actual operating conditions of structures, in particular the sequence of loading, installation, crack formation, etc.
5. Development of skills in applying modern computational and analytical programs for creating models of building structures and analysing calculation results.
6. Developing skills in working with design schemes for reinforced concrete, metal, bases and foundations, structures in complex engineering and geological conditions and in seismically active areas.
7. Introduction to the possibilities of applying computational intelligence (AI) in automated design and analysis of structures.

The importance of discipline

Mastering modern computer-based calculation methods is critically important for construction professionals, as it provides:

- high accuracy of stress-strain state analysis of structures;
- the ability to optimize project solutions;
- effective use of building materials;
- integration into digital design (BIM, digital twins, parametric modelling);
- adaptation to the requirements of the modern regulatory framework, which is actively evolving;
- ability to work across disciplines in project teams;
- competitiveness of specialists in the labour market in Ukraine and abroad.

Thus, the discipline provides a solid theoretical and practical foundation for design, research, and engineering activities in the field of construction.

1.2 Regulatory support for computer methods of calculating buildings and structures

Modern computer methods for calculating buildings and structures do not operate in a vacuum: their application is strictly regulated by both national and international regulatory acts that define acceptable methodologies, accuracy, interpretation of results, and even the structure of the calculation model. The correct use of these standards in the context of computer analysis is key to ensuring that projects meet safety, reliability, cost-effectiveness, and functionality requirements.

1.2.1 Principles of regulatory control of computer modelling

Regardless of the software used – whether it is LIRA-FEM, RFEM, ANSYS, SCAD, Abaqus, or other FEM complexes − the basic principles of structural modelling must comply with generally accepted requirements for:

- mechanical resistance and stability (safety & stability);
- limit states of the first and second groups;
- consideration of physical, geometric, and structural nonlinearity;
- taking into account the stages of installation, creep, crack formation, seismic loads;
- compliance of loads with applicable standards.

These principles are implemented in the software complex module, but the user is responsible for the correct formulation of the task, selection of elements, consideration of boundary conditions, and evaluation of results. Compliance with standards is not only a legal requirement but also a prerequisite for correct analysis.

1.2.2 Ukrainian regulatory framework

A. General requirements for calculations.

The basic document defining the principles for calculating buildings and structures in Ukraine is DBN V.1.2-14:2018 General principles for ensuring mechanical resistance and stability of structures – a document that establishes the methodological basis for the application of linear and nonlinear calculations, analysis of limit states, and provides references to the system of standards and norms that must be applied in design and modelling.

It is harmonized with Eurocode EN 1990, i.e., it is based on the same philosophical and methodological principles.

B. Standards for specific types of structures:

- DBN V.2.6-98:2020 – "Structures of buildings and structures. Concrete and reinforced concrete structures. Basic provisions" – establishes requirements for reinforcement, crack resistance, deformations, and elasticity modules, which directly affect the formulation of the problem in the FEA model;
- DBN V.2.6-198:2014 – “Metal structures. Design standards” – defines the criteria for strength, rigidity, and stability for steel elements, as well as the consideration of thin-walled, welded, and bolted connections, which must be correctly reflected in the design documentation;
- DBN V.2.1-10:2009 – “Bases and Foundations” – contains sections regulating the setting of the “base-foundation” interaction task in FEA analysis, including nonlinear soil deformability;
- DBN V.1.1-12:2014 – “Construction in seismic areas of Ukraine” – contains methods for calculating seismic effects that require the use of spectral methods or time integration – methods implemented in ANSYS, LIRA-FEM, and RFEM software;
- DBN V.1.2-2:2006 – “Loads and influences” – establishes normative values for all types of loads (dead weight, operational, snow, wind, seismic), which must be correctly specified in the design, taking into account the type of combination (partial, main, special).

C. Guidelines and methodological documents:

- DSTU-N B V.2.6-212:2016 – Guidelines for the design of reinforced concrete structures according to FEA, which contains recommendations on the selection of a calculation model, considering cracking, creep, and multilayered structures;

- DSTU B V.2.6-156:2010, DSTU B V.1.1-26:2010 – contain methods for substantiating crack resistance, calculating foundations and structures in complex soil conditions.

1.2.3 Eurocodes: the basis of modern design

Eurocodes are applied in Ukraine in parallel with national standards, especially in cases of design for international customers or work with software complex certified under Eurocodes (e.g., RFEM 6, SCIA Engineer, Robot). They are based on the same basic principles of limit states and the probabilistic approach.

Key documents:

- EN 1990:2002+A1:2005 – Basis of Structural Design – general design philosophy, including load combination principles;
- EN 1991 (1-1 to 1-4) – Actions on structures – wind, snow, and operational loads;
- EN 1992-1-1 – Concrete structures. General rules and rules for buildings;
- EN 1993-1-1 – Steel Structures. General rules;
- EN 1997-1 – Geotechnical design;
- EN 1998-1 – Design of structures for earthquake resistance.

These documents contain not only theoretical provisions, but also tables for calculating reliability coefficients, load distributions, soil parameters etc., which can be automatically applied in FEM complexes with the appropriate national annex.

1.2.4 Confirmation of compliance of software complexes with standards

Certification of software products for compliance with DBN or Eurocodes is carried out on the basis of official expertise. For example:

● LIRA-FEM – has a certificate of conformity with DSTU and DBN, as well as calculation modules in accordance with EN 1992 and EN 1993;

● RFEM 6 – has modules for automatic load combination generation according to EN 1990/1991 and wind/snow zone generation according to national annexes;

● ANSYS, Abaqus – require manual verification of compliance with standards, but can be used for specialized tasks (dynamics, impact, thermophysics) when properly aligned with standards.

1.2.4 Problems and recommendations

When teaching this subject, you should pay attention to these things:

● Developing an FEM model doesn't automatically mean it'll meet the standards – you need to check that the boundary conditions, stiffnesses, loads, and element types are all right. [1]

● The availability of "nonlinearity," "dynamics," and "geotechnics" modules does not eliminate the need to verify the adequacy of the task setting in accordance with the requirements of DBN V.1.2-14:2018 or EN 1990.

● The results of PC calculations must be interpreted in accordance with limit states – it is unacceptable to mechanically compare stresses with those not specified in the standards.

2 COMPUTER MODELING OF STRUCTURES AND THE FINITE ELEMENT METHOD

2.1 History and role of computer methods in civil engineering

The first attempts to use computer methods in construction appeared back in the 1950s, when computers and methods for solving systems of linear equations were being developed. The stiffness method, which is the basis of modern construction mechanics, made it possible to analyse bar structures manually, but as the tasks became more complex (plates, shells), the need for numerical methods became obvious. [2] The finite element method (FEM) emerged as a universal approach to solving differential equations in the early 1960s. Its idea is to discretize a complex area into small elements for which simple local equations can be written. The set of these equations forms a global system that can be easily solved on a computer. [3]

In the 1970s, the first commercial packages for engineering calculations (NASTRAN, ANSYS) appeared, and in the 1980s, specialized software complexes for builders (SAP90, LIRA) appeared. Currently, there are dozens of FEA programs on the market, ranging from general-purpose (Abaqus, ANSYS) to highly specialized (LIRA-FEM, RFEM 6, SCAD). Their popularity is explained by the fact that modern software allows not only to obtain the results of the stress-strain state, but also to automatically check the structure for compliance with codes, select cross-sections, perform reinforcement, and analyse dynamic and nonlinear effects. [4]

Modern structural analysis tools provide high accuracy, significantly reduce design time, enable the modelling of complex structures, and integrate with BIM systems. Such systems allow engineers to work with three-dimensional models, ensure the accuracy of calculations, automate code compliance checks, and help reduce errors. This is especially relevant for complex structures (bridges, high-rise buildings, industrial facilities), where manual methods are too labour-intensive.

The course "Modern Computer Methods for Calculating Building Structures" is aimed at developing competencies in the use of modern FEA packages, understanding the mathematical foundations of FEM, taking into account nonlinear and dynamic effects, as well as integration with BIM and AI systems. Key concepts are discussed below.

2.2 Idealization of the object when creating a model

The process of computer modelling of a structure involves building a simplified computational model that adequately reflects the operation of the real structure. To do this, the engineer needs to make a series of idealizations – deliberate simplifications and assumptions that allow them to focus on the main factors of strength and stability and filter out secondary details. Idealization is necessary due to the complexity of real structures: without it, mathematical calculations would be impossible to perform within a reasonable time frame. Properly performed idealization allows you to significantly simplify the model with almost no sacrifice in the accuracy of the results.

2.2.1Geometry idealization

The creation of a computational model begins with the idealization of the actual structure. It is necessary to determine the level of detail: for beams, frames, and trusses, it is advisable to use bar elements with two or three degrees of freedom at the node; for slabs, plate elements; for curved shells and tanks, shell elements. Software packages, such as LIRA-FEM and RFEM, have a large library of element types, ranging from linear bars to solid elements.

It is important to correctly reproduce the shape and dimensions of elements, select a coordinate system, and consider eccentricities. For example, when modelling floor beams, you need to specify the distance between nodes and take into account the width of the slab. For shell elements, ensure sufficient mesh density to describe the curvature; for complex geometries, use automatic mesh generation methods.

2.2.2 Idealization of boundary conditions

Real supports are replaced by hinges, clamps, or elastic connections. A hinged support allows rotation without transferring moments; a rigid clamp fixes all displacements and rotation angles; a movable support allows translational movement in a specific direction. In LIRA-FEM and RFEM, boundary conditions are specified at nodes by fixing degrees of freedom. Note that idealized supports can give significantly different results: for example, the critical load for a column with hinges is 4 times less than for a column with clamps. [1]

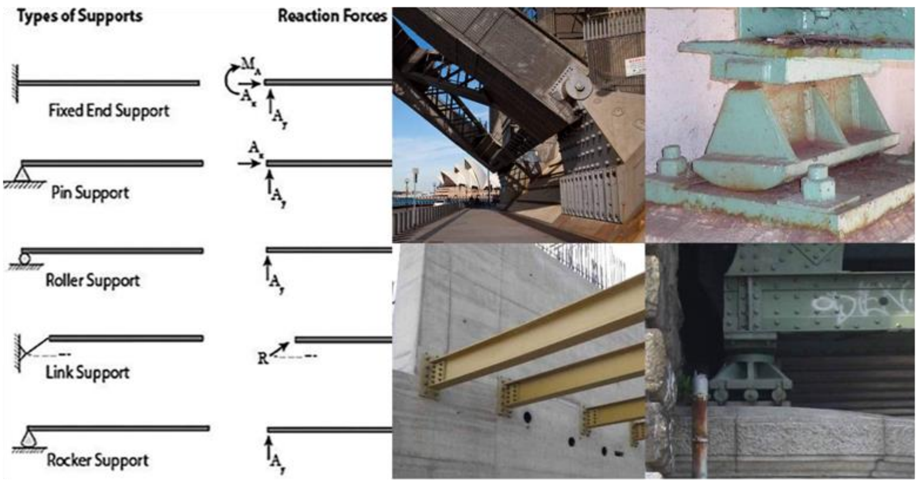

Figure 2.1 – Modelling and examples of supports in structures

Elastic connections are often used for foundations to simulate the compliance of the soil base. Software systems (SOIL in LIRA-FEM) allow calculating the bedding coefficient depending on the soil properties and foundation depth.

2.2.3 Idealization of loads

Loads and influences are of different nature – permanent (dead weight, soil pressure), temporary (live load, wind, snow), special (seismic, explosions). Simplifications are used in calculation models: distributed pressure is replaced by a point force or uniform load on an element, complex time laws of influence are represented by sets of static loads with coefficients.

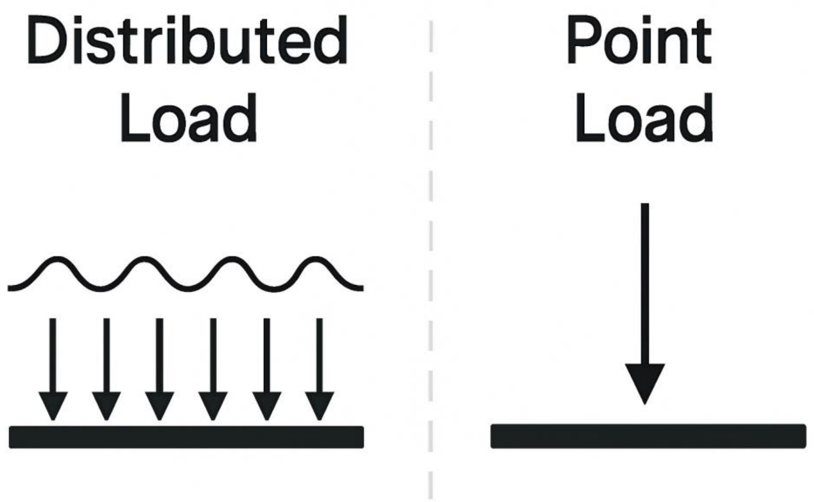

Figure 2.2 – Simplification of loads in calculation models

Modern programs have libraries of standard loads: for example, wind loads are specified by region and building height, and the program automatically distributes the pressure across the shell. In addition, standards require the formation of load combinations taking into account reliability and simultaneity factors.

2.2.4 Idealization of nodes and connections

The load-bearing frame of a building consists of elements (beams, columns, trusses, etc.) connected to each other at nodes. The actual behaviour of a node can be complex: a semi-rigid connection transmits torque partially, while a hinged connection transmits virtually no torque. In the calculation model, nodes are usually assumed to be either hinged or rigid. A hinged node in the model gives the elements freedom of relative rotation and does not transmit bending moments; a rigid node rigidly connects the elements, aligning their axes and transmitting the moment completely. Actual connections are almost always intermediate in rigidity, but the engineer must classify them for calculation. For example, in metal trusses, joints are assumed to be hinged (even if they are welded), based on the assumption that the truss only experiences axial forces in the members. This simplification is justified because the truss is designed so that the bending in the members is minimal. In contrast, in reinforced concrete frame structures, column-beam joints are modelled as rigid, since the monolithic connection ensures the transfer of bending moments. If it is known that the joint has a certain degree of flexibility, the models provide for the possibility of releasing moments – that is, introducing a hinge at the end of the element so that the moment is not transferred. This allows you to model semi-rigid connections, bringing the model closer to real-world performance, such as bolted joints on high-quality flanges.

In addition to the characteristics of the node itself, there are also design simplifications. Small details of nodes (overlay plates, stiffeners, gussets) are usually not modelled separately – their influence is taken into account in aggregate form.

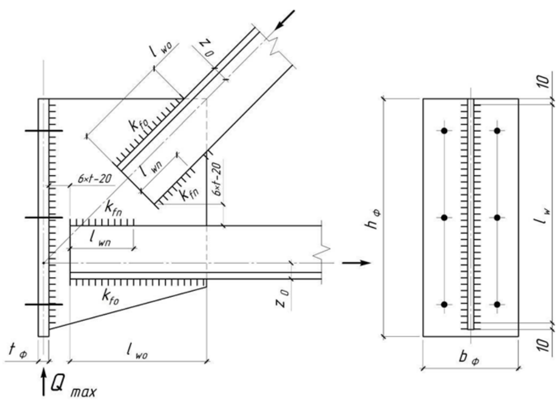

Figure 2.3 – Idealization of structural nodes in computational models

For example, the presence of stiffening ribs in a beam node is taken into account by increasing the moment of inertia of the cross-section or adding stiffness to the node, rather than by modelling each plate. Thus, the idealization of connections boils down to accepting a reasonable model of interaction between elements in nodes, which significantly affects the reliability of the prediction of the structure's behaviour.

2.2.5 Idealization of material properties

Another necessary simplification is the idealization of material properties. Real materials used in building structures – concrete, steel, brick, wood – have complex behaviour: they can be heterogeneous, anisotropic, and exhibit nonlinearity (plasticity, cracking, creep). To simplify the analysis in the calculation model, a homogeneous isotropic linear-elastic material is usually assumed. This means that the properties of the material (Young's modulus E, Poisson's ratio μ) are the same in all directions and do not depend on the stress level until the proportionality limit is reached. For example, reinforced concrete in calculations for the first group of limit states is often modelled as an equivalent isotropic material until cracks appear. Steel is assumed to be elastic to yield in most calculations.

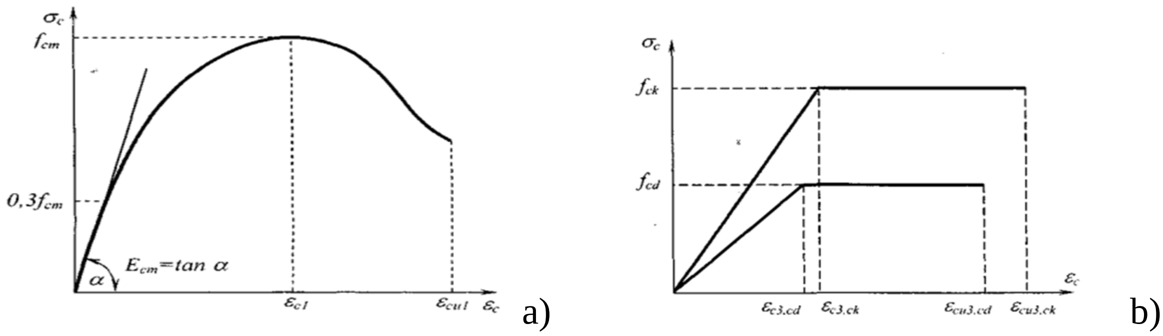

Figure 2.4 – Stress-strain diagrams for concrete: (a) linear-elastic stress-strain diagram for concrete; (b) bilinear stress-strain dependence for concrete

Linear elastic material idealization greatly simplifies calculations by allowing the application of the superposition principle. It is justified if the structure operates in the elastic stage – the stresses do not exceed the yield strength (for steel) or the crack formation limit (for concrete). When plastic deformation or failure is expected, the material model must be complicated by introducing multiline stress-strain diagrams, plasticity models, creep models, etc. Modern programs (e.g., Abaqus, ANSYS) offer libraries of nonlinear material models, including user-defined models. However, the use of such models requires high-quality experimental data for their calibration and significantly complicates the analysis. Therefore, in practical calculations, engineers often limit themselves to simpler idealizations. For example, the behaviour of steel reinforcement in reinforced concrete is modelled as ideally elastic-plastic: up to the yield point – elastic stage (modulus E), after – plastic flow at constant stress (modelling flow without strengthening). In strength calculations, concrete is assumed to be elastic until cracking occurs, after which its tensile stiffness is zeroed out – this is how crack opening is modelled. These simplified models comply with regulatory documents and have been tested in design practice.

2.2.6 Load combinations

The calculation is performed for many load cases. Standards (DBN V.1.2-2:2006, Eurocode 1) define combinations of permanent, long-term, short-term, and special loads. For example, the main combination when calculating the strength of reinforced concrete elements is 1,35G + 1,5Q, where G is the permanent load and Q is the short-term load. Software packages provide automatic formation of combinations and calculation in accordance with standards.

2.2.7 Typical simplifications of structural solutions

At the modelling stage, assumptions are also made about the structural design of the building. In particular, non-essential structural elements are often excluded from the model. Architectural details, finishes, non-load-bearing partitions – anything that does not affect the distribution of the main forces – are usually not modelled in order not to complicate the design. Such elements are taken into account indirectly if necessary (for example, the mass of non-load-bearing partitions is added to the permanent load on the floor, but the partitions themselves are not included in the frame model). The same is done with secondary details of load-bearing elements: a group of small elements can be aggregated into one equivalent. A striking example beams with through holes or composite steel-reinforced concrete slabs. Instead of modelling each beam, opening, and connection, a solid element with equivalent parameters (averaged stiffnesses) is introduced, which approximately reproduces the behaviour of the composite system. This preserves the load-bearing capacity, but the model becomes significantly simpler.

Modern software packages have libraries of typical elements for idealized modelling of complex assemblies and parts. For example, in LIRA-FEM or SCAD, you can add a hinge connection at the end of an element to the model with a single click – this is equivalent to a very flexible contact or plate insert, but the engineer does not need to model these parts explicitly. Such capabilities speed up model creation and make it easy to implement the necessary simplifications.

2.2.8 Model verification and reliability

After creating a calculation model, the engineer must ensure its adequacy. It is critically important to assess how the accepted idealizations have affected the results. It is recommended to perform verification calculations: for example, for a critical structure, two models can be compared – one simplified, the other more detailed – and the differences in forces and displacements can be examined. If the difference is insignificant, the simplified model is considered acceptable and used for design. In complex cases, the results of numerical modelling are compared with data from experiments or field tests – this is called model validation. In addition, the calculation complex itself must be verified – checked for correct operation on test tasks. Only when these conditions are met does computer modelling give reliable results.

In conclusion, it should be emphasized that any result of a computer calculation requires engineering analysis and common sense. If the input data or model is incorrect, the computer will return some result – but the responsibility for interpreting it lies with the engineer. Therefore, the modelling methodology includes not only building a model, but also critically evaluating the results obtained from the perspective of physical reality.

2.3 Typical problems of computer modelling

Despite the great capabilities of modern software, computer modelling is not infallible. There are a number of typical problems and sources of error that engineers should be aware of. They can be divided into three groups [5]:

● modelling errors arise due to simplifications and assumptions inherent in the model. A model never reproduces a real object 100 % accurately. Examples: incorrectly specified geometry or boundary conditions, distorted fixation (an important degree of freedom was not fixed, and the structure "floats"), incorrectly defined material parameters (for example, for isotropic steel $\mu = 0{,}5$, which makes the stiffness matrix degenerate)[5] , or incorrect type of calculation (for example, the dynamic nature of the problem or nonlinearity is ignored). Many of these errors are related to the human factor – inattention or insufficient understanding of the structure's operation. The motto "garbage in – garbage out" illustrates this problem well: if the model is set incorrectly, the software calculation will inevitably give an incorrect result. Therefore, it is important to check the input data, simplifications, and assumptions:

• discretization errors are related to the approximate nature of FEM calculations. They arise because a continuous structure is replaced by a finite mesh of elements. If the mesh is not fine enough, the results may have a significant error compared to the exact analytical solution. Usually, stresses and displacements converge to the correct values when the mesh is refined (the element size is reduced). The engineer should perform a convergence analysis: for example, reduce the size of the elements in the area of highest stresses and check how the calculated maximum stresses change. It often happens that a coarse mesh underestimates peak stresses and does not reveal local concentrations. On the other hand, an overly fine mesh increases the calculation time and the volume of results, although the gain in accuracy may be small. It is necessary to find a compromise level of discretization for each task. Modern programs include tools for assessing discretization errors and automatically thickening the mesh in "suspicious" areas, but the responsibility for ensuring sufficient mesh quality still lies with the engineer. Particular attention should be paid to singularities – points in the model where the stress theoretically tends to infinity (e.g., sharp corners, concentrated forces, or rigid fastenings of plate edges). In reality, such mathematical features do not exist (the load is always distributed over a small area, and the corners have chamfers), so infinite stresses are an artifact of the model. However, for the mesh, this means that when it is refined, the stresses will grow without limit, and the convergence criterion does not apply. Engineers need to be able to recognize singular zones and not interpret their "infinite" stresses directly. It is often sufficient to evaluate the stress not at the point of the feature itself, but at a small distance from it, where the value stabilizes to a physically meaningful level;

- numerical errors and limitations of algorithms are calculation errors associated with the limited bit width of a computer, the peculiarities of algorithms for solving equations, convergence conditions, etc. In most modern programs, they are minimal: stable algorithms with double precision numbers are used, automatic verification of Lagrange or Newton-Raphson conditions for nonlinear iterations, etc. However, sometimes the model may turn out to be poorly conditioned, and solving the system of equations leads to the accumulation of rounding errors. This manifests itself, for example, in the form of equilibrium inconsistencies or abnormal deformations. Such problems are more common in nonlinear dynamic calculations or when modelling contacts. The engineer should monitor the service messages of the software: if degenerated elements or a singular stiffness matrix cause the calculation to stop, the software package usually provides a warning. During nonlinear analysis, the program also controls the load step size and the number of iterations required to achieve convergence – if the model "fails to converge," it must be reviewed (whether the step size is too large or instability has occurred). Software tools are continuously being developed to overcome these difficulties, yet it is impossible to eliminate them completely.

Another problem of modelling is the interpretation of results. Computer analysis produces a vast array of numerical data (reactions, displacements, internal forces, stresses, etc. at thousands of points of the model). The engineer must be able to extract the essential information and draw conclusions for design. Here, two types of errors are possible: either overlooking a critical result (for example, a local peak moment in a beam), or, conversely, focusing excessively on a minor effect that does not influence the overall reliability. To avoid the first type of error, one should use the visualization and extremum search tools available in the software, and continually ask where the maximum stresses are expected, paying special attention to these areas. To avoid the second, it is important to remember the safety margin and the sensitivity of the structure: not every local stress concentration is destructive, and often a small structural measure is sufficient to reduce it (for example, adding a plate under a support force to distribute it).

Every computational model is only an approximation of reality and therefore has a specific domain of applicability. For example, if a linear-elastic material model is applied, the results are valid only up to the onset of plastic deformations. If the soil is modelled by elastic supports, the settlement prediction will be approximate, since real soil exhibits complex nonlinear behaviour. It is important for the engineer to recognize the limits within which the obtained results are valid and not to extrapolate them beyond these boundaries. If the problem extends beyond the capabilities of the model, it is necessary to switch to a more adequate model (for example, performing a nonlinear analysis instead of a linear one, or a full dynamic analysis instead of a static equivalent).

In summary, it should be emphasized that a significant share of errors in the process of computer modelling arises primarily not from the imperfection of the software, but from the human factor. Incorrect or incomplete user actions are among the main sources of inaccuracies. Therefore, the professional competence and attentiveness of the engineer during model development and input data analysis are the decisive prerequisites for obtaining reliable and trustworthy results.

2.4 Growth of Detail and Complexity of Models

Thanks to the progress of computational technology, engineers are now able to model ever deeper levels of structural behaviour. Whereas earlier models were predominantly global (the framework as a whole), it is now possible to account for local effects in detail. The trend toward increasing modelling depth is manifested in several aspects:

- multiphysics models. Modern software allows performing coupled analyses: for example, thermal history + stress (fire resistance calculation of a structure), hydrodynamics + mechanics (interaction of wind with a building or water flow with a bridge), electromagnetic effects + thermomechanics, and others. Such integrated models provide a comprehensive picture of structural behaviour under real conditions, where multiple factors act simultaneously;
- modelling of failure and damage. Numerical methods are being developed to account for the initiation and propagation of cracks (concrete fragmentation, ductile failure of steel) and for material degradation over time (reinforcement corrosion, metal fatigue). For example, in Abaqus, the Concrete Damage Plasticity models are implemented, allowing simulation of concrete damage accumulation and crack opening under cyclic loading. This significantly complicates the models but makes it possible to evaluate structural behaviour close to the ultimate limit state;
- Consideration of structural details. Whereas previously a frame joint was simplified as a hinge, it is now possible to explicitly model its components – such as plates, bolts, and welds – and analyze the stress distribution within the joint. For this purpose, submodells are used: the global model of the building provides the boundary forces, which are then applied to a local detailed model of the joint. In this way, a combination of broad coverage and detailed elaboration is achieved. Software such as IDEA StatiCa, which specializes in the analysis of steel joints by the finite element method with consideration of the actual geometry of elements and the plastic behaviour of steel, is gaining popularity. This allows you to design optimized (non-standard) nodes with knowledge of their actual strength reserve;
- "Structure-base" models. The modern approach to complex foundation conditions is the joint calculation of the structure and soil base (SSI – Soil-Structure Interaction). Software products are now available that allow you to build a finite element model of the soil around a structure or use boundary elements to model a

semi-infinite foundation. This way, it is possible to more accurately predict settlement, pressure distribution on the foundation, and the interaction of adjacent structures. For example, in Plaxis 3D, it is possible to completely simulate an excavation pit, pile foundation, and calculate the structure of the building together with the soil. Although this requires significant resources, for high-rise buildings or important engineering structures, this approach is justified to ensure the reliability of the forecast. [4]

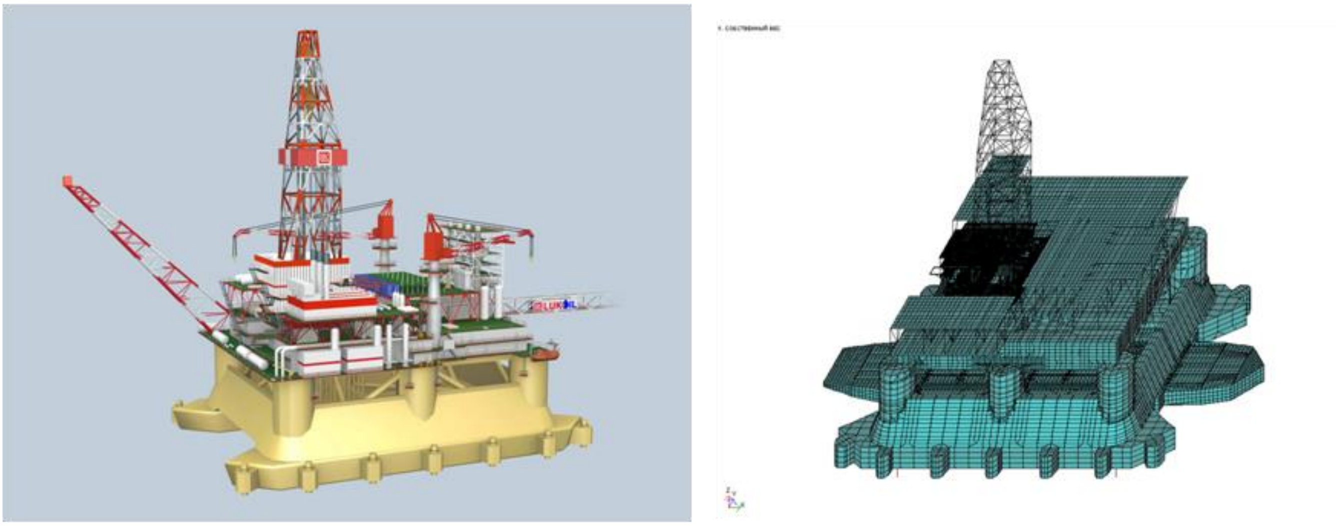

Figure 2.5 – Comparison of stress-strain state (SSS) for realistic and idealized models (Software package: LIRA-FEM)

2.5 Finite element method (FEM)

2.5.1 Analytical and numerical calculation methods

In classical analytical calculation, engineering problems are solved by deriving closed formulas based on differential equations of equilibrium and boundary conditions. Such approaches provide accurate solutions, but are often limited to simple models (linear materials, homogeneous geometry, simplified boundary conditions). For real complex structures, analytical methods become unsuitable or too labour-intensive. Instead, numerical methods (in particular, the finite element method, FEM) allow us to obtain an approximate solution through computational procedures. The numerical approach breaks down a complex problem into smaller subproblems and finds a solution iteratively or by solving a large system of equations, which is within the capabilities of modern computers. For example, the classic problem of determining the root of the equation $f(x) = 0$ may have an analytical solution for simple $f(x)$ (such as $x - 5 = 0$ gives $x = 5$), while for an arbitrary function, an engineer can apply a numerical method (bisection method, Newton's method, etc.), gradually approaching the solution. In general, analytical methods provide accurate formulas for idealized cases, while numerical methods provide engineers with flexible tools for calculating realistic structures with the required accuracy, controlling the approximation error. FEM occupies a central place among numerical methods, being the basis of most modern programs for calculating building structures.

2.5.2 Basic principles of the finite element method

The essence of FEM lies in the discretization of a continuous domain into a finite number of simpler subdomains – finite elements. The structure (or its computational model) is divided into a set of small elements of regular shape – in two-dimensional problems, these are usually triangles or quadrilaterals, and in three-dimensional problems – tetrahedrons, hexagons, prisms, or pyramids. The set of elements forms a finite element mesh that approximates the original geometry of the structure. Nodes are characteristic points of elements (corners of polygons, vertices of tetrahedrons, etc.) where the unknowns are concentrated, for example, the displacement of nodes in a building structure.

Interpolation through node values is used for an approximate description of the displacement field (or other desired function) inside an element. In other words, an unknown function (e.g., beam deflection, stress, temperature, etc.) is expressed as a linear combination of special basis functions – also called shape functions – with coefficients equal to the values of the function at the nodes[6]. These shape functions are known and selected in advance by the engineer (as a rule, these are low-degree polynomials that ensure the continuity of the field between neighbouring elements).

Thus, “node interpolation” means that the behaviour of an element is completely determined by the values of quantities at its nodes, and inside the element the field changes according to a simple law (linearly, quadratically, etc., depending on the choice of shape functions). For example, for a triangular element with linear shape functions, the displacement of any internal point is an interpolation of three node displacements:

$$u(x,y) = N_1(x,y)u_1 + N_2(x,y)u_2 + N_3(x,y)u_3,$$

where u_ii – the displacement of the nodes; $N_i(x,y)$ – linear shape functions that are equal to 1 in one node and 0 in others.

Once the form of the function inside each element has been specified, equilibrium equations are formulated for the elements and the entire structure. This can be done using variational principles (the principle of minimum potential energy) or weighted residual methods (the Galerkin method). In fact, substituting approximate functions into differential equations leads to a discrepancy (error), and the requirement for the extremum of this discrepancy gives a system of algebraic equations [7]. As a result, the entire calculation boils down to solving a large system of linear equations:

$$Ku = F,$$

where K – the so-called stiffness matrix of the structure, u is the vector of unknown nodal displacements (or other sought values); F – the vector of external influences (forces, loads) .

The stiffness matrix K is assembled (“collected”) by aggregating the contribution of each element – for this purpose, local stiffness matrices of elements are calculated based on selected shape functions and material properties, and then they are reduced to a global coordinate system and node numbering. The resulting matrix K is a sparse, symmetric, positive definite matrix of large dimensions (thousands or millions of unknowns for modern problems). Its dimension is equal to the number of degrees of freedom (DOF) in the system – the product of the number of nodes and the number of unknowns per node. By solving the system Ku = F (using direct or iterative methods of linear algebra), we obtain the displacement at all nodes of the structure. Based on the node displacements, the deformations of the elements (according to the dependencies of elasticity theory) and stresses are calculated – thus, the stress-strain state of the structure is determined.

The selection of basis (interpolation) functions is an important part of constructing an FEA model. The simplest choice is linear functions of the form (as in the triangle mentioned above), which gives linear elements with a constant gradient of values within the element. Such elements are quick and easy to calculate, but they do not give very accurate results if the actual field of values is significantly curved. To increase accuracy, high-order elements are used, with quadratic or cubic shape functions. Such elements have more nodes (for example, a second-order triangle has 6 nodes: 3 at the vertices and 1 on each side), correspondingly more degrees of freedom and more complex formulas, but they better approximate curved fields and allow for a larger mesh. Engineers are always looking for a compromise: higher-order elements provide accuracy on a coarser mesh, while lower-order elements require a finer mesh for the same accuracy. Modern programs offer elements of different orders, and users can choose them depending on the task at hand.

Boundary conditions (fixations, specified displacements, etc.) are entered into the FEM model by modifying the K matrix and the F vector – fixed degrees of freedom are removed or specified as known. In practice, this means that supports and hinges are modelled by corresponding conditions at the nodes: a fixed node will have zero displacements, a hinge will have zero rotation angles, etc. Other connections are modelled in the same way (elastic supports, contacts between elements – in a linear setting, they can be approximated as rigid or elastic linear connections between nodes). As a result of applying boundary conditions, the system of equations changes slightly (for example, rows and columns corresponding to fixed nodes are removed).

In summary, the finite element method algorithm is as follows:

1. Discretization of the structure. The engineer divides the structure into finite elements of a certain type (beams, plates, shells, solid elements – see below). The mesh topology is determined: which nodes connect which elements, where the loads are applied, which nodes are fixed.

2. Selection of shape functions. For each type of element, interpolation functions are specified that determine the distribution of displacements (or other unknowns) within the element through the node values.

3. Derivation of element equations. Local equations of an element, compiled according to the principle of minimum potential energy or by the residual method, are written in matrix form:

$$k^{(e)}\ u^{(e)} = f^{(e)},$$

where $k^{(e)}$– the stiffness matrix of the element; $u^{(e)}$– the vector of unknowns of the element; $f^{(e)}$– the local vector of loads on the element.

4. Formation of a global system. Local equations of all elements are summed up into a global system Ku = F. This uses the fact that neigh boring elements share common nodes (at the boundaries of elements, internal forces of equal magnitude and opposite direction act, which mutually compensate each other). The global matrix K has a dimension of $n \times n$, where n is the total number of structural elements.

5. Taking boundary conditions into account. The rows and columns of matrix K corresponding to rigidly fixed degrees of freedom are removed (or modified to take into account the specified displacements), and vector F is adjusted to take into account the reactive forces on the supports.

6. Solving the system of equations. Using a numerical algorithm (Gauss method, conjugate gradient method, etc.), we calculate the unknown vector u, thereby obtaining the displacement (or other unknowns) at all nodes of the structure.

7. Calculation of results. Secondary values are calculated based on node displacements: element deformations, stresses, reactions, stress state invariants (equivalent stresses, principal stresses), safety factors, etc., depending on requirements. At this stage, a post-computational error assessment can be applied – for example, energy balance analysis or integral error analysis – to determine how accurate the solution is and whether additional mesh refinement is necessary.

The finite element method guarantees convergence to the exact solution of a theoretical mechanics problem with gradual mesh refinement (or increasing element order). In practice, it is sufficient for an engineer to select a mesh such that the results obtained remain virtually unchanged when it is further refined – this indicates that the required accuracy has been achieved.

2.5.3 Types of finite elements and construction of a structural model

The type of finite element is determined by the shape of the element and the number of degrees of freedom it describes. In structural mechanics, the most common types of FEM are:

- 1-dimensional elements (bars): simulate longitudinal structural elements – beams, columns, trusses. The nodes of such elements have 6 degrees of freedom (3 displacements and 3 rotations in space) if it is a general beam element, or 3 degrees of freedom (only displacement in the plane) if it is a flat bar. Core elements come in different subtypes: beam (taking into account bending moment and transverse shear, as in a beam or frame), truss (only axial tension/compression, as in truss braces), bracket, cable (only tension without bending stiffness), etc. All of them boil down to elements based on two nodes. 1D elements are widely used for modelling building frames;
- 2-dimensional elements (plates and shells): simulate thin plate or shell surfaces – walls, floor slabs, thin shell coverings, tanks, thin-walled profiles. In two-dimensional elements, nodes usually have 6 degrees of freedom (in the general case of a spatial shell) or 3 degrees of freedom (in a flat setting). Subtypes: flat elements (work only on membrane forces in their plane – for modelling walls, stiffening diaphragms), bending plates (work on bending perpendicular to the plane – for floor slabs), shells (combine membrane and bending work, can be curved). Typically, 2D elements are triangular or quadrangular in shape;
- 3-dimensional elements (volumetric): simulate volumetric bodies of arbitrary shape – foundations, massive nodes, parts of complex geometry. These are tetrahedral or hexahedral elements, whose nodes have 3 translational degrees of freedom (rotations are not explicitly defined, because a solid body transmits bending

through the deformation of the layer of elements). Volumetric elements are the most versatile, but also the most resource-intensive (because a large number of elements are needed to fill the volume);

- specialized elements: these include soil foundation elements (elastic or elastoplastic foundations, which are sometimes modelled as separate elements such as springs or volumetric layers), contact elements (elements that provide friction/adhesion conditions between two surfaces), Hooke's rods (only longitudinal springs between two nodes), mass and damper elements (concentrated masses and damping connections for dynamic analysis), etc. Such elements are implemented to model special conditions, such as the operation of a structure on an elastic base (foundation on soil) or the modelling of local effects (hinge connection, damper etc.).

These elements can be used to build complex structural models. Modern software allows you to combine different types of elements in one model. For example, a building can be modelled using frame elements (beams and columns as rods), walls and slabs as shells, and the foundation as a volumetric array or elastic substrate. The RFEM (Dlubal) program supports the construction of such combined models: the main module allows you to define planar and spatial systems consisting of plates, walls, shells, and beams, and also provides the ability to model solid and contact elements. Similarly, LIRA-FEM has a library of elements of various dimensions, so engineers can create an optimal model by selecting the appropriate type of finite element for each part of the structure.

Choosing a model is a responsible task for an engineer. In the early stages of design, simplified models are often used: for example, a spatial frame for a building instead of a solid volumetric model, or a beam diagram instead of a detailed through-belt truss. Simplifications are justified if they do not distort the main results (strength, stiffness, stability). Guidance documents (e.g., DBN V.1.2-14:2018) require that the calculation scheme reflect the actual operating conditions of the object and its stress-strain state as accurately as possible. [8] This means that the engineer must take into account all factors that significantly affect the performance of the structure (e.g., interaction of elements, combination of different materials, possible concentrated masses or hinges). The same standard states that when forming a calculation model, it is advisable (where possible) to take into account nonlinear effects – geometric nonlinearity (the effect of deformations on equilibrium) and physical nonlinearity of materials. If nonlinear analysis is not explicitly required by the standards, it is permissible to assume a linear relationship between forces and displacements to simplify the calculation – of course, with subsequent verification of the strength of the cross-sections (for example, taking into account plastic strength reserves or crack formation, if such phenomena are expected). Thus, the standards give the engineer a certain freedom of modelling, but also oblige him to ensure that the model adequately describes the actual structure.

2.6 Finite element method (FEM)

2.6.1 Mathematical apparatus

The finite element method is based on the principle of minimum potential energy, according to which the equilibrium state of the system corresponds to the minimum value of the energy functional [9]. The solution domain is replaced by a discrete model of finite elements connected at nodes. For each element, shape functions are selected and a local stiffness matrix is determined. Next, local matrices are assembled into a global stiffness matrix using assembly rules. After taking boundary conditions into account, a system of linear algebraic equations is obtained:

$$Ku = F,$$

where K – the global stiffness matrix, u is the vector of unknown displacements; F – the vector of external forces.

The advantage of FEM is its ability to handle complex geometries, different materials, and boundary conditions. The CAE Assistant article explains that FEM breaks down complex problems into small, manageable elements, allowing the simulation of complex shapes, time-dependent loads, and materials with non-standard properties [10]. A separate advantage is versatility: the same procedures can be used for problems of elasticity, heat transfer, fluid flow etc.

2.6.2 Discretization and mesh generation

Discretization is a key stage of FEM. It determines the accuracy and speed of calculation. The mesh can be generated by the user (manual meshing) or by algorithms for automatic meshing of finite elements (Advancing Front, Delaunay). The choice of element type depends on the geometry: triangular and quadrilateral elements for plates, tetrahedral and cubic elements for spatial models.

Things to consider:

- mesh quality. Too large elements give rough results, too fine mesh increases calculation time. For areas with stress jumps (holes, concentrations), the mesh should be densified;
- mesh fitting. Special elements (adapters) or the “bandaging” technique are used in the gaps between different types of elements (e.g., rods and plates);
- approximation order. Higher order polynomials in shape functions provide more accurate results without grid refinement, but increase computational costs.

2.6.3 Types of finite elements

Various elements are implemented in the programs:

- linear (bar) elements. They model beams, columns, and trusses. They have 2-3 degrees of freedom at the node (translation, rotation);
- plate and shell elements. Used for slabs and thin-walled structures; take into account membrane and bending deformations;
- solid elements. Volumetric elements for modelling thick slabs, foundations, and arrays;
- special elements. Cable (wire rope) elements, soil foundation elements, prestressed elements.

Each type has its own limitations and areas of application; when selecting an element, it is important to consider thickness, aspect ratio, and the presence of bending moments.

2.6.4 System of equations and boundary conditions

After assembly, a system of equations $Ku = F$ is obtained. The solution requires consideration of boundary conditions: fixings, specified displacements, temperature effects. In linear problems, direct inversion or iteration methods are used; in nonlinear problems, iterative algorithms (Newton-Raphson, successive load application) are used. The results (displacements) are then used to determine the stresses and strains in the elements.

2.6.5 Advantages and limitations of FEM

Advantages:

- versatility and ability to model complex structures;
- integration with numerous physical processes (heat transfer, mass transfer, acoustics);
- automation of checks and optimizations;
- availability of a wide range of software products.

Limitations:

- the need for sufficiently accurate geometry and correct boundary conditions (incorrect idealization leads to false results);
- dependence on the quality of the mesh and the choice of element types;
- the complexity and computational costs of nonlinear and dynamic problems;
- the need for experience in interpreting results.

3 CALCULATIONS OF TYPICAL STRUCTURES IN ACCORDANCE WITH REGULATORY REQUIREMENTS

3.1 Linear formulation of the structural theory problem

A linear problem is a structural calculation that assumes proportionality between external loads and the corresponding structural response. In other words, linear modelling follows the principle of superposition: doubling all applied forces results in doubling all displacements and internal forces in the elements. Such direct proportionality is valid only under certain assumptions about the operation of the structure: (1) small deformations that do not change the geometric stiffness of the system; (2) linear elastic behaviour of the material (compliance with Hooke's law $\sigma = E\varepsilon$ until the elastic limit is reached); (3) constancy of boundary conditions (supports do not change their stiffness, no contact switching occurs, etc.). Under these conditions, the equilibrium equations of the structure are linear and can be written in matrix form $Ku = F$ with a constant stiffness matrix K.

The assumption of small deformations and displacements means that during loading of the structure, its initial geometry remains virtually unchanged. Therefore, the calculation equations can be compiled relative to the undeformed configuration, neglecting second-order effects (for example, the deflection of beams does not affect the calculation of the same deflections). This is the classic hypothesis of linear geometry, which cuts off geometric nonlinearity. For structures, it is usually assumed that deflections up to $L/250$ or $L/500$ (where L is the characteristic size) are small enough for this condition to be met. Otherwise, a geometrically nonlinear analysis must be applied.

Linear elasticity of a material means that the stress-strain diagram is a straight line up to a certain limit, and after the load is removed, the structure completely regains its shape. In terms of structural mechanics, we only work in the first stage of material behaviour, without reaching the yield point of steel or crack formation in concrete. All material nonlinearities (plastic deformation, cracks, creep, endurance limit) are not taken into account in linear calculations. Of course, when designing structures, strength verification is performed separately: by comparing the obtained stresses with the permissible ones (or using safety factors), which indirectly takes into account the elasticity limits. But the stress calculation itself assumes elastic behaviour, i. e., the absence of irreversible changes in the material structure.

The flat section hypothesis (Bernoulli) is another important assumption of linear structural theory, relevant for beam and plate elements. It states that a section that was flat and normal to the axis of the element before deformation remains flat and normal after deformation. This hypothesis underlies the classical theory of beam bending and plate theory, simplifying the analysis (transverse shear between layers is not taken into accounti. e., shear deformation is considered small). Bernoulli's hypothesis is valid for thin elements and small deflections; for thick plates or high-load beams, corrections may be applied (Timoshenko's theory with shear taken into account).

All these hypotheses and linear relationships allow us to apply the principle of superposition and solve problems in a linear setting. In practice, this means that the results from individual loads can be added together to obtain the results from their combination. The principle of superposition is fundamental: it is used both in classical analytical methods (superimposition of several load cases in beam calculations, Saint-Venant's principle, etc.) and in numerical methods (linear FEM). In software packages, this is implemented through a combination mechanism: for example, LIRA-FEM allows you to calculate the response to each elementary load separately, and then obtain results for any combination by linearly adding these private solutions. This approach is particularly useful when forming calculation situations according to regulatory requirements – load combinations DCL (design combinations of loads), DCF (design combinations of forces) can be easily obtained from the database of results for unit loads.

It should be emphasized that the linear model of the structure is correct only up to a certain load level. If the loads are too high, the structure enters a nonlinear mode (for example, plastic hinges appear in the frame or noticeable deflections of columns occur). Then the principle of superposition no longer works, and the equilibrium equations become nonlinear (additional terms appear in them, or the stiffness matrix K becomes dependent on displacements). Linear modelling can be considered as a first approximation when evaluating the performance of a structure under the assumptions of "small" loads and "elastic" behaviour of the structure. In most cases, this approximation is sufficient to ensure reliability: the structure is designed so that it does not enter the nonlinear operating range under standard loads (this is the essence of the concept of proportionality limits, or the use of safety factors).

3.2. The principle of superposition in structural analysis

As mentioned above, superposition is the basis of linear theory. Formally, the principle can be formulated as follows: for a linearly elastic structure, the effect of several independent loads is equal to the sum of the effects of each load separately. Within the framework of the FEM, this means that if the system $Ku_1 = F_1$ (reaction to the first load F_1) and $Ku_2 = F_2$ (reaction to the second load F_2) is solved, then for the total load $F_1 + F_2$, the solution will be $u = u_1 + u_2$. The global stiffness matrix K must remain unchanged – this is only possible if the condition of the structure (elasticity, geometry) does not change. The principle of superposition offers significant advantages in engineering calculations: it is possible to calculate the structure for basic simple loads (for example, separately for each type of load – permanent, temporary, wind, seismic), and then quickly obtain results for any combinations (various combinations of loads according to standards) by summing them up. This is done, for example, in the same LIRA-FEM software complex: first, a calculation is performed for each elementary load, and then the DCF (design combinations of forces) (calculated combination of forces) is formed by adding the resulting forces with the corresponding impact coefficients. This significantly saves time and allows for flexible consideration of the requirements of the standards for combining loads.

The principle of superposition applies not only to results (force, displacement), but also to the properties of the system as a whole. For example, dynamic properties (natural frequencies and vibration modes) in a linear formulation do not depend on the load level: they are determined only by the mass matrix M and stiffness matrix K.

If the structure is loaded and still remains in the elastic phase, its natural frequencies will not change. Similarly, the stability of the structure (critical loads) in linear theory is determined by a linear combination of initial forces, and if all these forces are multiplied by a certain factor, the critical value will simply change in scale.

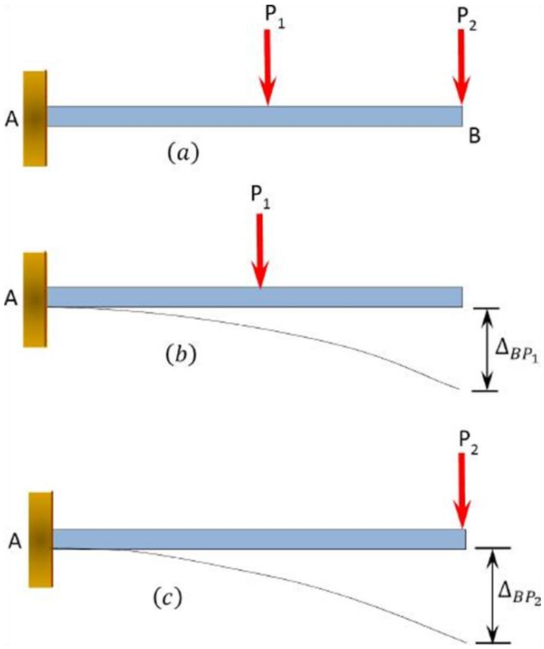

Figure 3.1 – Principle of superposition

Superposition is a consequence of the linearity of the system. If there are nonlinear effects in the structure (for example, stiffness changes when cracks appear, or forces are "cut off" when the yield strength is reached), then it is no longer possible to decompose the solution into partial solutions. Then each design case has to be modelled and solved separately (in a nonlinear setting, the combination of loads is also a nonlinear problem).

3.3 Basic assumptions of linear structural theory

Linear structural theory is based on several key assumptions:

1. Small displacements and deformations. It is assumed that all displacements of structural nodes are small enough to neglect changes in the geometry of the structure during calculation. The stiffness matrix K is formed for the initial, undeformed configuration and remains constant. This excludes the consideration of any geometric nonlinearities (second-order effects such as P–Δ

effects, column deflections in stability analysis, etc.).Formally, this hypothesis implies the use of a linear deformation tensor and a linear relationship between deformations and displacements. For most building structures under normal operating conditions, this condition is satisfied.

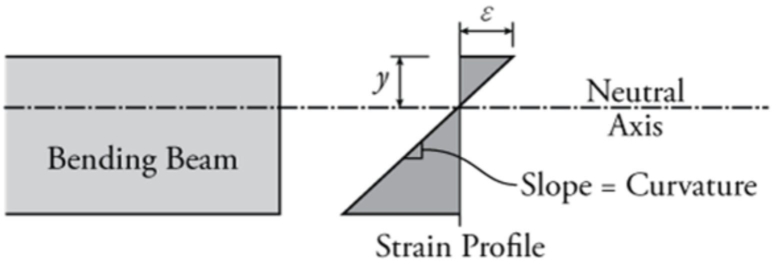

Figure 3.2 – Small displacements and deformations

2. Linear elasticity of materials. It is assumed that structural materials operate in the elastic deformation range, obeying Hooke's law – stresses are proportional to the corresponding strains: $\sigma = E \times \varepsilon$ for the elastic modulus E. The σ - ε diagram is linear up to the proportionality limit. Complete unloading leads to zero deformations (no residual deformations). This excludes the consideration of plastic phenomena, crack development etc. Again, this assumption is considered acceptable if the actual stresses do not exceed the elasticity limits of the materials with a margin (the margin itself is controlled by the selection of cross-sections and reinforcement).

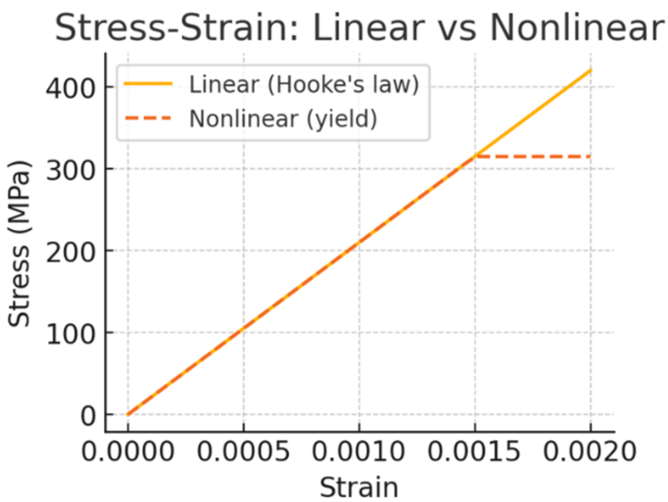

Figure 3.3 – Linear elasticity of a material

3. Hypothesis of flat cross-sections (for rods and plates). As mentioned, the cross-sections of a beam that were flat before deformation remain flat after

deformation (Bernoulli). In the case of plates, the normal to the median plane remains normal after bending (Kirchhoff's hypothesis for thin plates). This rejects shear deformations in the transverse direction and considers the distribution of deformations along the height of the cross-section to be linear. This hypothesis is valid for thin elements (relatively long/wide compared to thickness) with small bending curvatures. If the conditions exceed these limits (thick plates, compressed elements with significant bending), refined theories are applied (taking into account shear, as in Timoshenko's theory for beams, or in Reissner-Mindlin's theory for plates).

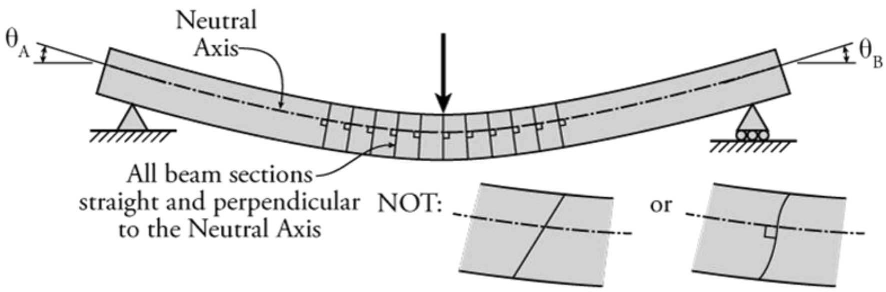

Figure 3.4 – Flat section hypothesis (Bernoulli)

These assumptions form the basis of classical formulas in structural mechanics. When they are satisfied, the behaviour of the structure is described by linear equations, and the calculation task is reduced to linear algebra (as described in the section on FEM). Deviations from these assumptions require a transition to more complex (nonlinear) models, which will be discussed below in the context of comparing linear and nonlinear formulations.

3.4 Setting up a calculation model in linear analysis

Building a calculation model is the first stage of structural analysis. In linear analysis, the principles of model building are the same as those for FEM in general. However, there are some features that should be noted:

1) selection of degrees of freedom. In linear calculation, you can take advantage of the fact that the object of study is inextricably linked to its initial configuration. For example, you can divide a spatial model into separate planes if the

loads and supports are symmetrical (the structure operates in a plane stress state). This is called the formation of flat or axisymmetric diagrams. Linearity ensures that out-of-plane deformations will not appear “by themselves,” so we can discard unnecessary internal forces. Similarly, in a linear static problem, inertial internal forces (no acceleration) or hydrodynamic interactions (if the structure is not in contact with liquid, etc.) can be ignored;

2) modelling boundary conditions. In linear analysis, boundary constraints (supports, hinges, elastic supports) are modelled as ideally rigid or ideally hinged. This is a simplification: for example, a real support on the ground has finite stiffness (the ground is flexible), but in linear calculations, it is often assumed that the end of the rod is completely rigid. Such a simplification is usually safe (somewhat conservative: a completely rigid clamped support gives more moments in the beam span than a slightly compliant one actually does). If accuracy is required, elastic elements with a given stiffness are introduced (for example, the soil bed coefficient in a foundation slab model);

3) linear materials. The model uses material constants (elastic modulus E, shear modulus G, Poisson's ratio nu) – usually from standard reference books (such as E_b = 3 × 104 MPa for B30 concrete, E_s = 2 × 105 MPa for steel etc.). They do not change during the calculation. The effects of creep or cracking of concrete are not taken into account in a purely linear model; if they need to be evaluated, this is done separately (for example, by reducing the equivalent E for cracked areas, or by modelling elastic-creep properties in modified linear analyses);

4) loads and combinations. In a linear problem, different types of loads (permanent, temporary, special) are introduced as separate cases and then combined by superposition. For example, regulatory documents (DBN, Eurocodes) contain requirements for load combinations – in linear analysis, these are simply linear combinations of several design cases, which are performed automatically. It should be ensured that the applied load does not change the structure of the system (the absence, for example, of loads that can cause displacement of supports or a change in the nature of contact is a separate complex topic). Usually, in linear analysis, we are

free to apply any forces or displacements without worrying that they will "break" the model – the model remains linear and stable for calculation up to infinite loads (the results will also grow proportionally to infinity, but the solution will converge). This distinguishes linear formulation from nonlinear, where too high a load can lead to calculation failure (divergence);

5) hecking limit states. After obtaining the results of linear calculations – stress, force, displacement – the engineer checks them for compliance with criteria (strength, stiffness, stability) in accordance with applicable standards. If any criteria are not met, the design should be changed (increase cross-sections, add elements) and the calculation repeated. Since the analysis is linear, such iterations in the selection of cross-sections are fairly straightforward: the results can be scaled (e.g., if the stresses exceed the permissible values by 20 %, the cross-sectional area can be increased by approximately the same 20 %). In a nonlinear setting, there is no such linear proportionality, so the selection requires trial and error with repeated recalculations.

An example of constructing a calculation scheme: suppose we have a building frame with a change in level in part of the plan (half-floor). The model can be constructed as a spatial frame with node connections, taking into account the half-floor through corresponding elements of different lengths. The figure below (conditionally) shows the general view of such a building and the layout of the half-floor elements:

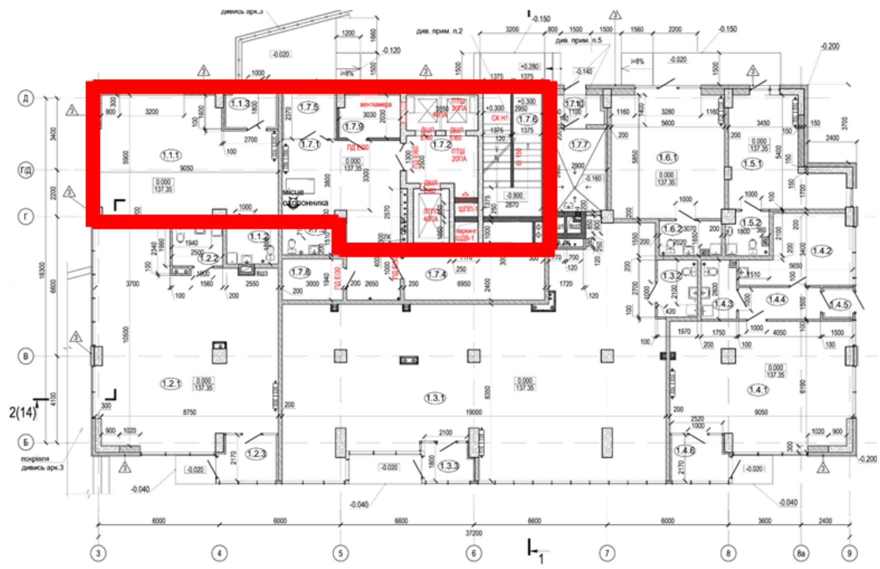

Figure 3.5 – Building plan with the level difference zone (half-floor) highlighted, which requires special modelling in the calculation scheme

Engineering-geological conditions are also taken into account: for example, you can specify elastic column supports, the coefficient of the bed under the foundation slab, etc. For demonstration purposes, illustration 3.6 shows a fragment of an engineering-geological section and a diagram of the location of wells for exploration:

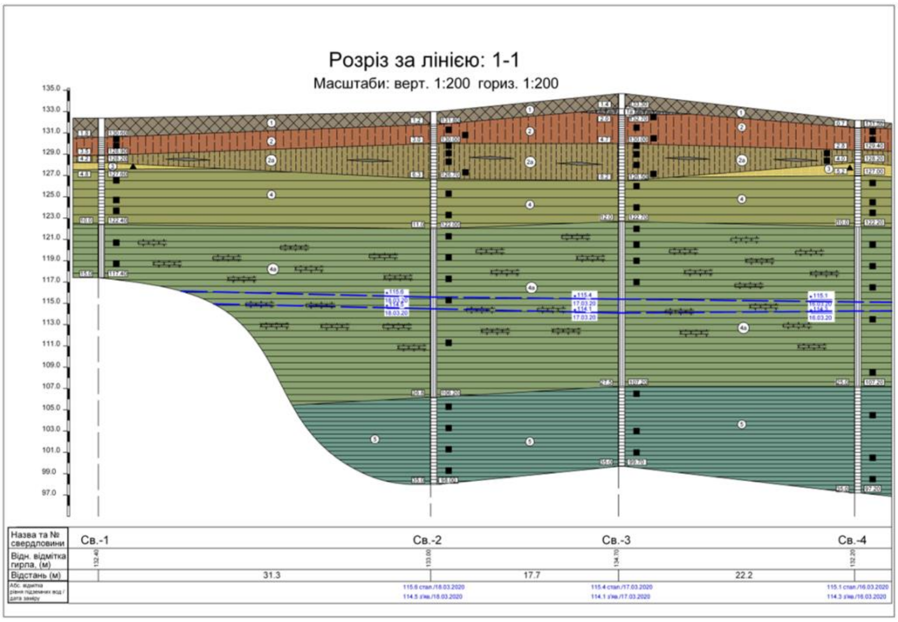

Figure 3.6 –Engineering-geological section (soil layer diagram) and location of test wells for the facility – these data serve as the basis for modelling the foundation of the structure in the calculation scheme

In linear calculations, the foundation is most often modelled in a simplified manner (either with the ends of the columns clamped rigidly or with elastic connections based on the soil deformation modulus). This is again a compromise between accuracy and simplicity, which is acceptable at the preliminary design stage. If necessary, the model can always be detailed – up to a complete three-dimensional model of the soil massif, but this goes beyond the scope of linear formulation and belongs to complex nonlinear problems.

3.5 Simplified dynamic calculation of structures (linear dynamics)

Dynamic analysis in a linear setting considers the vibrations of structures under the influence of dynamic loads (e.g., seismic activity, wind, shock pulses) or under initial excitation (natural vibrations after the application of an initial impulse). A simplified approach in linear dynamics is to use the principle of superposition in the time domain: to decompose the complex motion of a structure into the sum of its “natural” vibration modes. To do this, first determine the natural frequencies fi and vibration modes ϕi of the structure (vibration modes) – solve the eigenvalue problem for the pair of matrices K (stiffness) and M (mass). This problem boils down to solving the eigenvalue equation:

$$(K - \omega^2 M)\ \Phi = 0,$$

Where $\omega_i = 2\pi f_i$– angular frequencies; Φ – the matrix of eigenmodes (by columns – vectors Φ_i).

As a result, we obtain a set of eigenfrequencies f_1, f_2, ...,fn and corresponding normalized modes Φ_1, Φ_2, ..., Φ_n. The number of modes n here is equal to the number of degrees of freedom of the system (but in practice, the first few dozen modes are significant, while higher modes have very small effects and are often neglected).

3.5.1 Spectral analysis method (method of natural vibrations)

After finding the natural forms, the dynamic load (e.g., seismic impact) is decomposed according to these forms. According to the principle of superposition, the response of the structure can be presented as the sum $u(t) = \sum r_i(t)\phi_i$ – the time coordinate (generalized displacement) along the i-th mode. The problem boils down to finding the functions $r_i(t)$ – for a linear system, this is the solution of simple differential equations (one for each degree of freedom) with frequency ϕ_i. As a result, the complex process of vibrations is replaced by the summation of several harmonic components. The spectral method implements this approach in the frequency domain: knowing the soil response spectrum (seismic spectrum) or the applied load spectrum, it is possible to determine the response amplitudes for each natural mode and then combine them to obtain a complete picture of the structure's response. For example, according to the DBN V.1.1-12 standards for seismic areas, the soil response spectrum is specified as the dependence of the $K_{seismic}$ coefficient on the oscillation period. Knowing the first frequency of frame vibrations (let's say, $f_1 \approx 3$ Hz i.e.,

$T_1 \approx 0{,}33$ s), the corresponding response value is taken from this spectrum and the calculated inertial loads on the structure, the resultant from seismic activity, are determined. The computer performs this procedure automatically: the "Seismic analysis (linear-spectral)" module in LIRA-FEM allows you to enter the normative spectrum and obtain results in the form of maximum floor displacements, accelerations, and inertial forces in the elements.

3.5.2 D'Alembert's direct integral (numerical integration of equations of motion)

This is an alternative approach in which the system of equations of motion $M\ddot{u}(t) + C\dot{u}(t) + Ku(t) = F(t)$ is integrated step by step in time. For a linear system, effective algorithms (Newmark method, Wilson-θ method, etc.) with constant matrices M, C, K can be applied. The time step is selected based on the highest frequency that needs to be accurately taken into account. As a result, time dependencies of all desired quantities are obtained – displacements, velocities, accelerations, forces. This method is more versatile (it is also suitable for nonlinear systems, as discussed below), but it is often unnecessary for linear systems, since the spectral method can be used without forming a complete signal in time. However, direct integration is necessary if the load is not harmonic and is not covered by the standard spectrum (e. g., explosive action, impact). LIRA-FEM has a "Dynamics over time" module that allows you to solve both linear and nonlinear dynamics problems, providing tools for accurate and flexible modelling of any dynamic influences.

The results of linear dynamic analysis usually include the maximum values of the sought quantities (amplitude response) and vibration characteristics. For structures, in particular, the following are determined: maximum displacements and accelerations of each floor (important for assessing comfort and equipment integrity), internal forces in elements from inertial loads (to check strength), distribution of mass and transverse forces along the height of the structure, etc. Figure. 3.7 shows an example of free vibrations of a single-mass oscillator after an impulse disturbance:

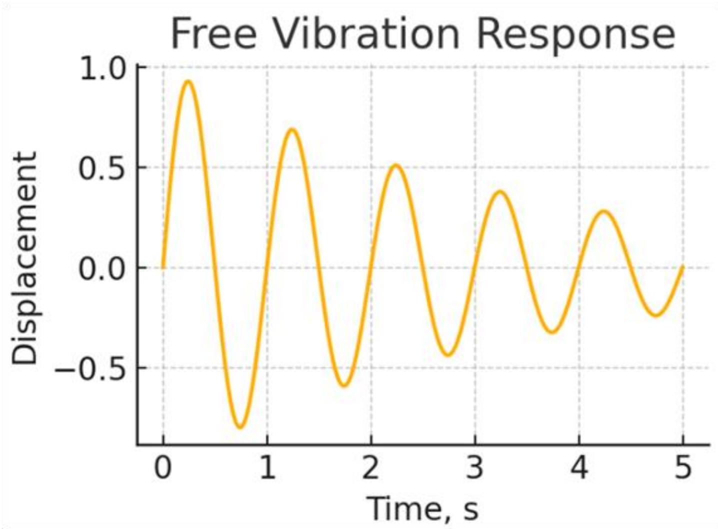

Figure 3.7 –Example of damped free oscillations of a single-mass system after an impulse (linear model with damping) – oscillations occur at the natural frequency of the system and the amplitude gradually decreases due to damping

For a real building, damping (viscous or other) can also be taken into account in the form of a C matrix. In linear analysis, it is customary to specify a certain percentage of critical damping (5 % for reinforced concrete, 2–3 % for steel – typical values) to estimate the attenuation of vibrations. The graph shows that when damping is taken into account, the vibrations decay, i.e., the displacement tends to zero over time. If damping is not considered (conservative case), then the natural vibrations in a linear elastic system will continue indefinitely (in reality, there is always at least a small amount of decay).

3.5.3 Scope of application of linear dynamics

Linear dynamic analysis provides reliable results as long as the structure remains intact and elastic during vibrations. For most scenarios (e.g., seismic activity at the design level according to standards), this assumption is acceptable – the structure is designed with a margin to prevent damage from design earthquakes. However, if an extreme impact or emergency situation is considered, where plastic deformation, crack development, and loss of stability of elements are possible, linear analysis will not be able to take this into account. For example, during a very strong earthquake, cracks may appear in reinforced concrete walls, causing the stiffness of the walls to decrease and the load distribution to change. A linear model will not detect this because it assumes constant stiffness. Therefore, in such cases, a nonlinear dynamic analysis is required (more on this below). However, for the vast majority of design cases, linear dynamics is sufficient and much simpler to perform. It serves as a useful and relatively quick tool for assessing the behavior of a structure under short-term influences (vibrations). Modern software packages (such as LIRA-FEM, RFEM, SAP2000, Abaqus/Standard, etc.) have effective algorithms for calculating natural modes and spectral analysis, so engineers can easily obtain the dynamic characteristics of a structure at the preliminary design stage.

3.6 Linear calculation of stability (loss of stability)

One of the key issues in structural theory is the stability of a structure's equilibrium. Loss of stability (buckling) is a phenomenon whereby, upon reaching a certain critical load, an elastic system transitions from one state of equilibrium to another (deformed) state without a proportional increase in load, often leading to destruction. A classic example is a rod under axial compression that suddenly bends (bulges) at a critical force P_{cr} (Euler's formula).

In linear analysis, stability is assessed by finding the eigenvalues of the stability problem. Consider a structure that has been previously subjected to a certain static load F_0 (which creates forces in the elements). The stiffness matrix K corresponds to elastic forces, but in the presence of compressive forces, the elements have a lower ability to withstand additional loads – the so-called geometric (compressive) flexibility appears. This is taken into account by the so-called geometric stiffness matrix K_g, which is proportional to the magnitude of the internal compressive stresses. Critical loss of stability is formulated by the condition that the effective stiffness of the structure degenerates:

$$(K + \lambda_{cr} K_g)\, d = 0,$$

where d – the eigenvector of displacements (the form of lateral bulging);λ_{cr}– the parameter (eigenvalue) corresponding to the critical load level [13].

This is an eigenvalue problem, similar to the eigenvalue problem, but here K_g replaces the mass matrix. Hence, λ_{cr} determines the scale by which the initial stresses (or initial load F_0) must be increased in order for instability to occur. The result is often presented as a critical load factor: $n_{cr} = \lambda_{cr}$, then the critical load,$F_{cr} = n_{cr} \cdot F_0$. For example, if $n_{cr} = 3,5$, this means that with a 3,5-fold increase in the current load, the system will lose stability. If $n_{cr} < 1,0$, it means that the system is unstable at the existing load (in practice, it should have collapsed earlier, or the model is incorrect) [11].n_{cr} values greater than 1,0 are interpreted as a stability reserve.

Based on this, linear stability analysis in software complexes (so-called eigenvalue buckling analysis) is performed in two stages: (1) a conventional linear calculation of the structure is performed for a given base load F_0; (2) for the obtained state (membrane forces are taken into account), the above-mentioned equation is solved, and the first few eigenvalues λ_i and the corresponding buckling shapes d_i are found. A_s a result, the program outputs the critical load coefficients and modes of stability loss. For example, LIRA-FEM has a special type of calculation “ANV – stability analysis” for such a case; similarly, RFEM (Dlubal) contains both linear stability analysis and a nonlinear stability analysis module (more on this below) [12].

3.6.1 Physical meaning of eigenvalues of stability

Essentially, λ_{cr} is a coefficient by which the current internal forces can be multiplied to achieve neutral stability. In this case, the structure can move in the direction of its own shape d without additional energy input. The corresponding own shape usually shows the configuration of the bulge. It is important to understand that formally linear stability analysis assumes that the structure has reached this state without previous lateral deviations. In reality, however, the slightest disturbances (slight curvature of the element, eccentricity of the load) lead to noticeable lateral deflections when the load is less than F_{cr}. Therefore, linear analysis somewhat overestimates stability – it gives a theoretical upper estimate of F_{cr}, which in reality cannot be achieved due to initial imperfections. Nevertheless, this result is very useful: it is used as an indicator of potentially unstable forms. An engineer, seeing that, for example, the second natural form – local bulging of the beam wall – has $n_{cr} = 4{,}2$, understands that with a 4,2-fold increase in load, local loss of stability is possible and this node should be checked in more detail. In design practice, there are requirements that the stability reserve factor be no less than a certain value (usually 3–5). If n_{cr} is less than this, stability must be increased: add bracing, increase the thickness of the element etc.

Euler's formula for an ideal compressed rod is a special case of the above method (which was known long before the advent of FEM). For a hinged rod, the critical force is Pcr $= \frac{\pi^2 El}{L^2}$. From the point of view of self-analysis, here $\lambda_{\text{cr}} = \frac{Pcr}{P0}$ if P_0 is the current load. This formula can be used to roughly check the correctness of the FEM results: if the model is a single hinged rod, the program should output $n_{cr} = \frac{\pi^2 EI}{NL^2}$ (where N is the compressive force, N corresponds to P_0). If this is the case, then the element matrices have been formed correctly.

3.6.2 Limitations of linear stability analysis.

As already mentioned, this analysis only considers membrane forces as a factor of instability. [11] That is, only axial forces (in plates – internal plane stresses) are included in the K_g matrix. This is called the first-order P-Δ effect. The influence of moments (rotations, deflections to bulging) is not taken into account, as is physical nonlinearity (material plasticity). Therefore, linear stability analysis usually overestimates critical loads. For structures operating in a zone of geometrically unstable equilibrium (e.g., thin-walled shells, geodesic domes), a more complex geometrically nonlinear stability analysis must be used, in which the structure is loaded gradually (incrementally) and the load level at which the algorithm cannot find equilibrium (the moment of loss of stability) is tracked.This approach is implemented in the form of nonlinear calculations with imperfections (for example, RFEM has a Nonlinear Buckling module, or Abaqus can perform Riks analysis). However, it is much more complex and sometimes requires the introduction of initial shape defects.

Therefore, linear stability analysis is a quick way to estimate potential buckling shapes and approximate critical load factors. It is widely used in practice as part of the second limit state (loss of stability) calculation. According to the standards, if the structure has a sufficient margin based on the results of linear analysis (n_{cr} above the threshold), a more detailed nonlinear analysis may not be required.

3.7 Example of Linear Structural Analysis and Comparison with a Nonlinear Formulation

Let us briefly consider a real example and demonstrate the difference between the results of linear and nonlinear analysis. Example: a five-story reinforced concrete frame building with shear wall diaphragms subjected to seismic loading. A linear spectral analysis (performed using the LIRA-FEM program) shows maximum displacements at the top of the building of 12 mm, with no element exceeding the elastic limit (strength utilization factors ≤ 0,9). The stability coefficient n_{cr} for the most compressed column equals 4,8, which corresponds to a stability reserve of approximately 480 %. At first glance, the structure appears to meet all requirements.

However, let us perform a physically nonlinear analysis: we take into account the stress–strain diagrams of concrete and reinforcement and model the formation of plastic hinges at the ends of the beams. At the same time, the applied load remains the same design load. The result: the maximum displacements at the top of the building increased to 20 mm (due to the “softening” of the frame after reinforcement yielded in certain joints), and plastic hinges formed in two columns (i.e., they reached the ultimate plastic rotational deformation). Although the building withstood the load (no unlimited growth of deformations occurred), part of the elements entered the plastic state. The linear analysis did not reveal this, instead showing lower force values.

This example illustrates a general fact: linear analysis does not account for the redistribution of internal forces once the material exceeds the elastic limit, and therefore may somewhat underestimate deformations and overstressing in certain zones. At the same time, the linear approach in such a case provides a safe result in terms of strength – it “did not capture” the redistribution of moments, but due to the safety factors, the structure nevertheless proved to be adequate.

Thus, when a structure operates close to or beyond the elastic limit, the linear model becomes insufficient. Modern design codes (for example, DBN V.1.2-14:2018, clause 6.1.3) require in such cases the performance of more advanced analyses or verifications. In particular, for structures of a high consequence class, it is necessary to carry out verification calculations using different models, including those that account for the physical nonlinearity of materials [8].

What nonlinear analysis provides that linear analysis does not: it makes it possible to account for plastic deformations, cracking, stiffness changes under loading, and contact phenomena. Numerically, this is carried out using iterative methods – the most well-known being the Newton – Raphson method, which is implemented in most software packages (such as Abaqus, Ansys, etc.) [13].Its essence lies in the successive refinement of the approximation to the equilibrium state of the structure through the linearization of nonlinear equations at each step. In this process, it is often necessary to update (recalculate) the stiffness matrix after each iteration, since it depends on the current state (for example, after yielding of an element, its local stiffness decreases, and this must be reflected in the system matrix) [13]. Nonlinear analysis requires more time and may fail to converge if the model is poorly defined (for this reason, for example, failure states are often modelled by introducing small initial imperfections; otherwise, a perfectly symmetrical stability model will simply not initiate movement – it needs a trigger for bifurcation).

In Abaqus and other professional systems, the solution of a linear problem is regarded as a special case of a nonlinear one – the so-called linear (elastic) behaviour is considered a subset of nonlinear behaviour. The software allows combining both approaches: for example, it is possible to perform a linear dynamic analysis based on the state obtained after a nonlinear static calculation (the perturbation eigenvalue method – linear perturbation analysis in Abaqus) [14]. This is useful when it is necessary to determine the vibration modes of an already cracked or plastically deformed structure. However, such sophisticated cases go beyond the scope of typical design analysis.

In summary, the linear formulation of problems is the first and most common step in the analysis of a building structure. It provides the engineer with the essential information on internal forces and displacements, allows for the selection of cross-sections, and enables strength verification. Nonlinear analysis represents the next level of detail, applied when necessary (for example, if linear analysis indicates insufficient load-bearing capacity, or if the structure by its nature operates in the nonlinear phase – such as in seismic design with ensured plastic hinging). The two approaches do not contradict but rather complement each other. Often, a linear analysis is performed first, and its results are then used to identify zones where nonlinear effects may potentially occur (areas of plastic deformations, loss of stability), which are subsequently analyzed locally in a nonlinear manner. This approach makes it possible to optimize the workload.

3.8 Potential Energy of Deformation and Its Role in Analysis

One of the fundamental concepts of mechanics is the notion of the potential energy of a deformed system. For an elastic structure, this energy consists of the strain energy (stored in the material due to the work of internal forces) and the potential energy of external forces (the work of external loads). The principle of minimum potential energy states that an elastic system in a state of stable equilibrium possesses the minimum total potential energy [3]. In other words, the increment (variation) of the total energy equals zero for small perturbations of the configuration in equilibrium. This principle is an alternative form of expressing the equilibrium equations and underlies calculation methods, in particular the Finite Element Method (FEM).

When constructing an FEM model, one can apply a variational approach: formulate the functional of the total potential energy Pi(u) of the structure (the difference between the strain energy U and the work of external forces W on the displacements u), and then require $\delta\pi = 0$ for any small variations δu of the admissible displacement field. This leads to the same set of equations Ku = F. Thus, mathematically, FEM is equivalent to the requirement of stationarity (minimum) of potential energy [8]. This fact is important not only theoretically but also practically: it guarantees that the solution obtained by FEM is the most energetically favourable (among other possible finite element approximations). However, if nonlinearities or inconsistencies appear in the model, the potential energy may lose convexity, and multiple stationary points may arise (alternative equilibrium configurations, as in stability problems).

In the context of linear calculation, it is important to know that deformation energy can serve as a measure of stiffness. For example, if a structure is very flexible, it accumulates little elastic energy under a given load (most of the work is done by external forces), and conversely, a stiff structure accumulates more elastic energy (less work is done by forces). Castigliano's principle, for example, states that the displacement in the direction of the applied force is equal to the derivative of the deformation energy with respect to that force. This allows deflections to be calculated using energy relationships.

In stability analysis, the minimum potential energy criterion provides a convenient way to determine the moment of instability: at critical load, the potential energy has a flat minimum (neutral equilibrium state), and when exceeded, the minimum turns into a saddle point, and the system rolls down to another minimum (post-critical state). This requires a nonlinear approach to study, but, again, linear analysis can predict this moment – namely, when the determinant of the stiffness matrix $K+\lambda K_g$ becomes zero (eigenvalue), this corresponds to the second variation of the potential energy in a given direction = 0 (neutral stability).

In summary, the concept of potential deformation energy is useful to engineers for intuitively understanding the reliability of a structure. In linear analysis, it is possible to calculate the total elastic energy accumulated in the structure under the design load. If we add the potential of external forces (with a minus sign) to it, we obtain the Pi functional. Its minimum is achieved in our solution. This guarantees the uniqueness and stability of the solution to the linear problem. If, when modifying the structure (for example, during optimization), the deformation energy decreases, then the structure becomes more economical (less material is used to withstand the load). Such considerations are used in structural optimization methods, where the objective function is the elastic energy or total potential energy.

Program calculation reports often include an energy balance check – total internal energy = work of external forces. For linear elastic calculations, this is always true (with numerical error accuracy). If not, it is a sign of an error (incorrect model or convergence). Energy methods also underlie the assessment of FEA error: for example, the Page's criterion or Z2 criterion takes into account residual stresses, which can be interpreted through energy inconsistency.

3.9 Types of engineering nonlinearity in structures

Nonlinear behaviour of a structure can be caused by various physical and geometric factors. From an engineering point of view, there are several main types of nonlinearities: physical (material), geometric, structural, and genetic nonlinearity. Each of these types represents a separate aspect of deviation from linear behaviour, and they often occur simultaneously. Let's take a closer look at them.

3.9.1 Physical nonlinearity (material nonlinearity)

Physical (material) nonlinearity means that there is no direct proportionality between stresses and strains in the material. In other words, after exceeding the elastic limit, the material behaves inelastically: additional deformations increase nonlinearly with increasing stresses. For most structural materials, this manifests itself as plastic and other nonlinear deformations. [15] For example, low-carbon steel, after reaching the yield point, exhibits a horizontal plastic stress plateau, followed by material strengthening; concrete, on the other hand, has a nonlinear stress-strain diagram from the very beginning of loading (there is no clear region of perfectly elastic behaviour).Thus, the modulus of elasticity E ceases to be constant, and the stiffness of the elements decreases with the development of plastic or crack deformations in the material. It is also important that the deformation diagram may be asymmetrical: for example, concrete has different compressive and tensile strengths, so its behaviour under tension includes the formation of cracks long before the compressive strength is reached.

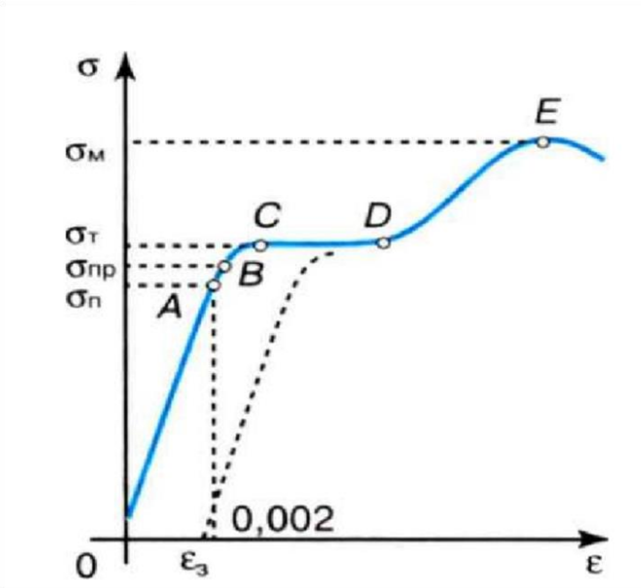

Figure 3.8 – Stress-strain diagram of a typical metal (low-carbon steel)

A typical stress-strain engineering diagram for low-carbon steel, illustrating the physical nonlinearity of the material. First, there is a linear-elastic stage (straight line section), followed by a horizontal yield section (yield point), and then a strengthening stage until the strength limit is reached. After the stress peak, the material loses its load-bearing capacity (necking, failure). Such curves show that after the elastic linear stage, the σ(ε) dependence becomes nonlinear.

Physical nonlinearity can be mathematically accounted for through nonlinear material laws – for example, by specifying deformation curves for steel and concrete. In numerical analysis (finite element method, FEM), this is implemented either through iterative recalculation of effective elastic moduli or by using plastic models (ideal plastic flow model, with strengthening, etc.). The solution of a physically nonlinear problem is usually performed using an iterative step method: the applied load is divided into small increments; at each step, the stresses in the elements are calculated for the current load level, based on which the stiffness characteristics (elastic moduli) are refined and the equilibrium is checked. Iterations continue until convergence is achieved – the difference between external and internal forces must become small. This approach takes into account the gradual decrease in the stiffness of elements during plastic deformation of the material.

For reinforced concrete elements, physical nonlinearity means taking into account the behaviour of concrete after cracking (when concrete has almost no tensile strength) and the plasticity of reinforcement. In practical programs such as LIRA-FEM or Ansys, corresponding material models are implemented: for example, the "Concrete-21" or "Concrete-22" models in LIRA-FEM provide a nonlinear concrete deformation diagram, while the material types "C245-21" or "C345-21" provide two-line steel yield diagrams (taking into account the yield strength). This allows for more realistic deflections and force distribution in reinforced concrete elements, including the redistribution of moments after the formation of plastic hinges etc.

3.9.2 Geometric nonlinearity (shape nonlinearity)

Geometric nonlinearity is associated with changes in the geometry of a structure under significant displacements and deformations. In linear analysis, it is usually assumed that changes in the shape of the structure do not affect the distribution of forces (small parameter). However, if the deflections or rotations are large, the actual (deformed) scheme differs from the initial one, and this changes the conditions of equilibrium. A classic example is the $P - \Delta$ effect in a compressed column: when a thin column deflects, the vertical load P creates an additional bending moment $M_{add} = P \cdot \Delta$ (where Δ is the deflection), which further increases the deflection and moment (second-order effect). If these secondary effects are not taken into account, the forces and deformations may be underestimated. It is generally assumed that for beams, deflection $< L/250$ (where L is the span) is small enough to neglect geometric nonlinearity. However, for larger deflections, especially in flexible frames and tall structures, a geometrically nonlinear calculation is required. For example, for a cantilever column with a large axial force and lateral load, a nonlinear calculation can give a 30–40 % higher moment at the base than a linear calculation.

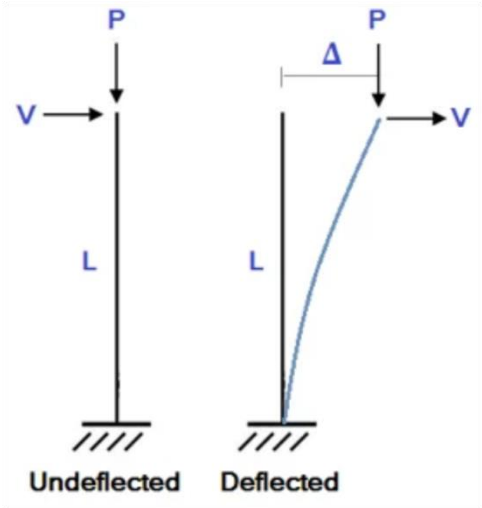

Figure 3.9 – Figure of a cantilever column under the simultaneous action of vertical load P and lateral force V in a non-curved (left) and curved (right) state

The solution of a geometrically nonlinear problem is performed using step-by-step methods. Special algorithms are implemented in FEM programs: the large displacement method, the rotation approach, etc., which at each step recalculate the stiffness and internal forces taking into account the current deformed geometry of the elements. In essence, a nonlinear equilibrium system $K(u)u = F$ is formed and solved, where the stiffness matrix K depends on the displacements u (due to the changed geometry). A critical case of geometric nonlinearity is loss of stability – a situation where, upon reaching a certain load, equilibrium in a straight line is impossible (the load-displacement curve has a maximum). Special algorithms are used to analyze post-critical behaviour, such as the arch length method (more details below).

3.9.3 Structural nonlinearity

Structural nonlinearity occurs when the design scheme of a structure changes during deformation. In other words, connections or supports change depending on the magnitude of displacements or loads. This is typical for so-called contact problems or situations with broken connections. An example is the deflection of a flexible beam that initially rests on two supports and has a gap in the middle to a third intermediate support. If the deflection increases and the beam touches the intermediate support, the scheme suddenly becomes three-span instead of two-span. Another example is the destruction of elements or jamming of hinges: when the force exceeds a certain threshold, a connection failure may occur (for example, an anchor bolt slips out or an insert crumples and no longer transmits the load). This also changes the static scheme of the system.

Modelling structural nonlinearity requires special techniques. Contact elements or nonlinear connections are introduced into the calculation models: for example, elastic supports with limited strength or elements that are activated only when nodes are joined. During step-by-step calculation, contact or strength conditions are checked at each step: if they are met, the corresponding connection is enabled or disabled. Thus, the topology of the model may change during the analysis. This is a rather complex type of nonlinearity that requires almost logical conditions in the solution process, so algorithms must provide for a correct response to changes in the scheme (restarting iterations when the state of connections changes, etc.). Examples: modelling the joint of two plates through contact with a friction coefficient; modelling supports on soil that can break away (the soil works only in compression – "one-sided" support), modelling the work of struts that act only in compression and lose their effectiveness in tension etc.

3.9.4 Genetic nonlinearity (pedigree)

A special type of structural nonlinearity is the so-called genetic nonlinearity, which is related to the history of loading and construction of the structure. The term "genetic" (from the word genesis) emphasizes that it refers to the sequential accumulation of stresses and deformations during the construction process. Another name for it is pedigree nonlinearity. In principle, genetic nonlinearity is a special case of structural nonlinearity: at different stages of construction, the structure has a different layout (not all elements have been installed yet) and a different initial stress state, so the final state differs from what it would be if the entire system were loaded simultaneously.

For example, let's imagine the construction of a multi-story frame floor by floor. The lower columns initially bear their own weight and the load of the first floor, possibly undergoing some plastic deformation or cracking. Then, when the next floors are added, these columns are additionally loaded with new weights and operational loads. As a result, their stress-strain state (SSS) will be different than if the entire building had been loaded with the full load from top to bottom at once. Thus, the history of load application and element installation affects the distribution of stresses.

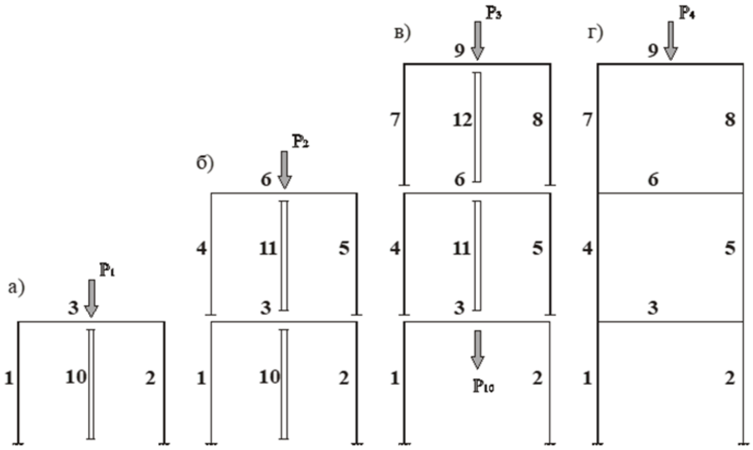

Figure 3.10 – Genetic nonlinearity (Source: LIRA-FEM)

To account for genetic nonlinearity, a construction sequence analysis is used. In the program, this is implemented by dividing the model into stages: elements and loads are entered in parts, stress relaxation from concrete creep can be performed between stages, material hardening is taken into account, etc. Well-known examples include the calculation of large-span bridges by phased suspension of spans, where a new section of the bridge is added at each step; the calculation of high-rise buildings taking into account load P-Δ effect errors during construction, etc. Stage analysis allows you to identify secondary stresses from the sequence of installation and more accurately estimate the accumulated deformations (for example, uneven shrinkage or deflection of slabs during the phased concreting of different sections). This approach is mandatory for unique structures and high-responsibility structures, where neglecting genetic nonlinearity could lead to errors in stress estimation and, accordingly, to the risk of damage in actual operation.

3.10 Engineering nonlinearity – a simplified calculation method

As noted, a full-fledged physically nonlinear calculation can be labour-intensive and not always justified for typical structures. In such cases, the approach of so-called engineering nonlinearity is used – this is a method of approximate consideration of nonlinear behaviour without explicit nonlinear analysis of the entire system. The essence of this approach is to estimate the effective reduction in the stiffness of structural elements (beams, columns, slabs) due to cracking, plastic deformation, and other nonlinear effects, and then perform a conventional linear calculation with these reduced stiffness values.

This method is particularly common for reinforced concrete structures, since a complete physically nonlinear calculation of reinforced concrete structures is complex. For example, the LIRA-FEM complex implements an engineering nonlinearity algorithm consisting of the following steps:

1. A basic linear calculation of the structure is performed with the initial values of the stiffness of the elements.
2. Based on the results of internal forces, the program automatically determines the required reinforcement in reinforced concrete elements (or the user specifies it from the outset).
3. Based on the obtained forces and selected reinforcement, new inertia moments and modules are evaluated for each reinforced concrete cross-section: that is, reduced stiffness taking into account cracks and plasticity. For example, beams are modelled by elements of variable stiffness, slabs – by orthotropic shells with reduced bending stiffness [16].
4. The structure is recalculated with the updated stiffness parameters. If necessary, steps 3–4 are iterated until convergence is achieved (usually 1–2 iterations are sufficient).

As a result, we obtain a distribution of forces and displacements closer to the nonlinear analogue, although the calculation formally remained linear. This approach makes it possible to take into account the redistribution of moments in reinforced concrete after the formation of cracks and a decrease in its stiffness, without explicitly introducing cracks into the model. According to the developers of LIRA-FEM, the use of engineering nonlinearity increases the calculated reinforcement requirement by only ~3–4% compared to a conventional linear calculation [26]. This means that the structure is slightly less economical, but more reliable (because real nonlinear effects are taken into account). The engineering nonlinearity method is also enshrined in regulatory documents: for example, DBN V.2.6-98:2009 provides for a reduction in the calculated stiffness of reinforced concrete elements in operational (second limit) stages to take into account concrete cracking.

3.10.1 Mathematical formulation of nonlinear problems

The transition from linear to nonlinear formulation means that the system of equilibrium equations of the structure becomes nonlinear. In general terms, the equilibrium equation can be written as:

$$R(u) = 0,$$

where $R(u) = K(u)u - F = 0$ (here $K(u)$ is a stiffness matrix that depends on the displacements u (due to changes in material properties or geometry) and F is a vector of external forces).

In the linear case, K is constant and does not depend on u, so the solution is trivial: $u = K^{-1}F$. In the nonlinear case, there is no analytical solution – it is necessary to find u and K simultaneously, often by iteration. Nonlinear equilibrium equations may have several solutions or no solutions at all at certain load levels (corresponding to a loss of stability).

In practice, nonlinear analysis is performed by searching for the equilibrium path of the structure. This can be imagined as tracing the load-displacement (or other parameter) relationship along a curve, which is usually inelastic. It is necessary to increase the load step by step and find a new equilibrium state at each step, satisfying the equation $R(u) = 0$. In the area where the curve rises, the system has one solution for each load; when the peak load (critical point) is reached, the standard method may "go off the branch," so there are special algorithms (e.g., the arc length method) that allow you to track the falling branch of the curve (post-critical path).

Convergence criteria are an important component of mathematical formulation: it is necessary to determine when the iterative process should be stopped. As a rule, the discrepancy is controlled by forces ($\|R(u)\|$ should become very small) or by displacements (the change in $R(u) = K(u)u – F = 0$ between iterations should be small). The tolerance for all discrepancy norms is set by the user or automatically by the program, based on the required accuracy.

3.10.2 Methods for solving nonlinear problems

A number of numerical methods have been developed for solving nonlinear equilibrium equations. They can be divided into two groups:

1. Iterative methods – involve repeated refinement of the solution until convergence, but it is not known in advance how many iterations will be required.
2. Direct methods – the calculation is performed in a fixed number of steps without internal iterations; accuracy is not guaranteed, but is often sufficient.

The most common iterative algorithm is the Newton-Raphson method. It consists of the following: for the current approximation $u^{(k)}$, the discrepancy $R(u^{(k)})$ and the corresponding correction Δu are calculated from the linearization system $K(u^{(k)})\Delta u = -R(u^{(k)})$. This correction is added: $u^{(k+1)} = u^{(k)} + \Delta u$, and the process is repeated [18]. Newton's method provides quadratic convergence speed near the correct solution, i.e., it very quickly "approaches" the exact value if the initial approximation is good enough. The disadvantage is that it is necessary to form and solve a system of equations at each iteration, which is computationally expensive. In some modifications (Newton-Kantorovich method), the stiffness matrix is not updated at each iteration, but less frequently or not at all, which reduces the cost of computation at the expense of slower convergence.

Another iterative approach is the method of successive approximations with variable stiffness (secant). It is essentially a simplified Newton method: after each iteration, the stiffness of the elements is updated based on the obtained state (for example, E is reduced for cracked concrete, etc.) and the linear system is solved again. Such iterations can be interpreted as a cycle: linear calculation – stiffness update – recalculation – …, until the results stabilize. This method has the advantage of simplicity, but it can get stuck if the stiffness reduction is too strong at some step.

Step analysis is one of the direct methods. The simplest version of this is the sequential loading method: we break down the external load into a series of small increments Δλ (for example, 10 %, 20 %, ..., 100 % of the full load) and for each step we solve a linear equilibrium problem, assuming the stiffness to be constant and equal to the initial value. This gives us a sequence of states that approximately lie on a nonlinear curve. The method is very simple, but errors can accumulate because we do not correct the solution as we go (there are no iterations for each step). An improved version is the step method with consideration of inconsistencies: after each step, the resulting inconsistency (unbalanced forces) is analyzed and its effect is added to the next step, thereby partially taking into account the nonlinearity. This increases accuracy with a slight increase in complexity.

The arch length method deserves special mention. It belongs to the direct methods, but controls not the load increment, but the increment along the equilibrium curve (in the “load-displacement” space). In essence, an additional condition is introduced that links the increments of loads and displacements so that the total step is constant (the length of the step along the curve). This method allows you to pass critical points of the curve when $\frac{\partial F}{\partial u} = 0$ (peaks, troughs), which the usual step method cannot do. The arc-length algorithm is widely used in the analysis of stability loss, structural collapse, etc., where there are falling segments on the F(u) diagram (for example, after reaching the strength limit of concrete, the element “falls” in bearing capacity – the curve goes down). The Ricks method also allows these states to be found.

In practice, modern software packages usually combine approaches. For example, LIRA-FEM, Ansys, Abaqus, etc. use the Newton-Raphson step-by-step iterative scheme: the load is applied incrementally, and at each step several Newton iterations are applied to accurately reach equilibrium. This provides reliability (each step converges) and error control, and also allows you to adjust the load step: if the iterations converge poorly, the program will automatically reduce the step $\Delta\lambda$. This ensures a balance between performance and accuracy.

3.10.3 Modelling of loading and erection processes

As noted in the section on genetic nonlinearity, the sequence of loading and erection stages can significantly affect the results. Therefore, in tasks where this is important, staged modelling of the loading and erection process is used. This effectively adds another level of cycles to the calculation: in addition to internal nonlinearity iterations, external cycles are performed in stages.

LIRA-FEM, RFEM, Midas, and others have tools for stage analysis. The user specifies several design states (stages), each of which includes its own model configuration and loads. For example, stage 1: the lower 5 floors of the frame are erected and loaded with their own weight; stage 2: the next 5 floors are added, and the load is increased; stage 3: the entire frame is completed, and operational loads are added etc. The program calculates them sequentially, transferring the accumulated stresses and deformations to the next stages. At the same time, creep between stages can be taken into account: for example, during construction, the lower floors of concrete are partially deformed by creep, and this removes some stresses or increases deflections. LIRA-FEM has options for setting creep and shrinkage coefficients for such stage calculations.

Load modelling can also be dynamic (not in time, but in magnitude): for example, applying snow load in an increasing layer – you can break it down into 5 steps and track when a plastic hinge appears in the frame. Or for a bridge: gradually increase the weight of the vehicle to find the load limit.

3.11 Practical examples of nonlinear analysis in software

Let us briefly consider several examples of how modern software complexes perform nonlinear calculations. Let us take a multi-story reinforced concrete frame modelled in LIRA-FEM. The columns and beams are defined as reinforced concrete bars with nonlinear material diagrams: concrete of the BN model (nonlinear-elastic until failure under compression deformation ε_{cu}), reinforcement – elastic-plastic with a yield strength. Loads include dead weight, live load on floors, and wind load. The program first performs a linear calculation for these loads, then, at the user's request, a physically nonlinear calculation: with a step of, for example, 10% of the load. In the first steps, the entire structure is elastic; as the deformations increase, cracks appear in the stretched areas of the beams (the program takes this into account by reducing the stiffness of the cross-sections), and the reinforcement reaches yield at the bases of some columns (a plastic hinge is formed there). After several steps, the load reaches the design maximum – further solutions do not converge, which indicates the limit state (the bearing capacity is exhausted). Analysis results: redistribution of moments – moments partially transferred from beams to columns due to column plasticity; deflections increased (compared to linear) by ~20% due to cracks; reinforcement in columns increased, especially in lower floors, to ensure plastic durability. This example demonstrates that nonlinear analysis allows us to see potential failure mechanisms: in this case, the formation of plastic hinges in columns, which can lead to brittle failure. If the structure proved to be too sensitive (simultaneous failure of several columns), structural reinforcement would be proposed.

Another example is a metal frame structure designed in RFEM 6 (Dlubal). Metal is modelled as elastic-plastic (for a profile, for example, steel with a yield strength of ~355 MPa). The load is a seismic shock, modelled by equivalent static forces. It turned out that several truss members were subject to yielding. The program performs a geometrically nonlinear calculation taking into account P–Δ (which is important for a high frame). Result: the frame withstands the load but shows significant residual deformations. Particular attention is paid to the local stability of elements: RFEM checks the wall panels of beams to ensure that they do not buckle locally (according to EN 1993-1-1). If the stability factor is > 1 somewhere, this means that reinforcement (stiffening ribs) is required. Thus, the comprehensive nonlinear analysis includes strength (limit state group 1), serviceability (group 2), and stability, all of which are interrelated.

The Ansys program allows you to simulate more complex nonlinear phenomena, such as contacts between elements (structural nonlinearity) or large deflection of shells considering changes in shape (geometric nonlinearity). For example, a disc-shaped shell (cover) with a central hole, which is supported along the circumference, was simulated. Under high pressure, the shell deflects so much that the edge in the center rises and breaks away from the support – this is a contact problem. Ansys uses special contact elements with a "bracket" option that respond only to compression. The calculation showed that after the detachment, part of the load was redistributed to other supports, and additional membrane forces (dome effect) arose in the material. This scenario is difficult to consider linearly, but nonlinear analysis gave engineers an understanding that the structure is safe even when one of the supports breaks away, because there is sufficient strength reserve (the material entered the plastic region but did not fail).

3.12 General principles of constructing calculation schemes

Before delving into the details, let us recall the basic approaches to creating a calculation model. Simplifying the actual structure into a calculation scheme is the first step in any analysis. The engineer must decide which elements to model explicitly, and which can be taken into account indirectly (through loads or changed parameters). The choice of a spatial or flat model is also critical: if the structure is loaded mainly in one plane (for example, a frame that carries only vertical and one direction of horizontal loads), a flat model with 2D elements can be used. For more complex cases (building frames with multidirectional loads, the presence of floor slabs, etc.), a full-fledged 3D model is required.

Example: a two-span building frame. If wind loads act only in its plane, a flat frame model is allowed (bar elements with 3 degrees of freedom in the node: 2 displacements + rotation). In this case, the main dimensions – column height H, spans L – are specified, and the support of the columns on the foundation is modelled by appropriate fastenings (hinged or rigid). If the frame is not isolated in space (for example, there are connections between adjacent frames, spatial covering), it will be necessary to model part of the system spatially or to take these connections into account separately.

Degrees of freedom (DOF): When creating a new task in software programs, you are often asked to select the model type based on the number of degrees of freedom in a node (for example, “type 2 – three DOF (flat scheme)” or “type 6 – six DOF (spatial)” in LIRA-FEM). This determines what movements the elements can have. For flat frames, there are 3 DF: horizontal and vertical displacement of the node, plus rotation (around an axis perpendicular to the frame plane). For spatial frames, there are 6 DF (three displacement components + three rotations). Incorrect selection of degrees of freedom can lead to unnecessary degrees (for example, if you set 6 degrees of freedom, but the structure is actually flat, there will be parasitic zero modes) or to the closure of real degrees (if 3 degrees of freedom for spatial – out-of-plane deformations are lost).

In addition to selecting a geometric scheme, it is important to identify the types of structural elements. In MSE programs, the main types are: rods (beams, columns, trusses), plates/shells (for slabs, walls, shells), arrays (solids) for volumetric areas (foundations, blocks). In practice, for building structures, massive elements are rarely modelled as solids (except for special zones, such as nodes or foundation soil); rods and plates are more commonly used. For example, reinforced concrete frames are often modelled only with bar elements for columns and beams, and floor slabs are modelled with plate elements (finite plate elements) or even as an equivalent load on the beams if simplification is required.

3.13 Features of modelling reinforced concrete structures

Reinforced concrete structures combine concrete and reinforcement that work together: concrete mainly absorbs compression, while reinforcement absorbs tension. This imposes certain requirements on the model in order to adequately reproduce this joint work. Key points of modelling reinforced concrete structures:

- types of elements and consideration of nodes. Columns and beams (girders) of the frame are modelled as rods of a given cross-section. It is important to consider rigid inserts in the nodes: a real column-girder node has a certain height (to the lower edge of the girder, for example). In order for the model to correctly convey the bending moment, it is often advisable to insert short rods of rigid material into the node (analogous to a rigid core) or to specify the eccentricities of the elements. [18] This will ensure the correct angle of rotation and interaction. Floor slabs, if modelled explicitly, are represented as plate-like (2D) elements connected to the beams along the contour. To make the slab and beam work together, either the aforementioned rigid inserts or the combination of nodes across the entire width of the slab (rigid diaphragm) are used;
- slab modelling. There are two approaches: 1) explicit modelling of slabs with plates/shells (for example, element type 40 in LIRA-FEM, or Shell in SAP2000), 2) indirect consideration – specifying the load from the slab on the beams without an explicit element. The first approach is more accurate, the second is simplified. In explicit slab modelling, its thickness, material (usually concrete), and the directions of the working reinforcement (e.g., as the orthotropy of the material) are specified. Distributed loads (from partitions, temporary loads) are applied directly to the plates and are independently redistributed to the beams through the stiffness of the slab. It is also important to take into account the openings in the slabs (they are modelled by cutting out part of the finite element mesh) and the possible hinged connection of the slab to the beam if it is hinged. As a rule, a monolithic floor works in conjunction with a beam, i.e., a rigid connection must be provided at the junction (this is achieved either by joining the slab and beam nodes or by using a “rigid joint” model). [17]

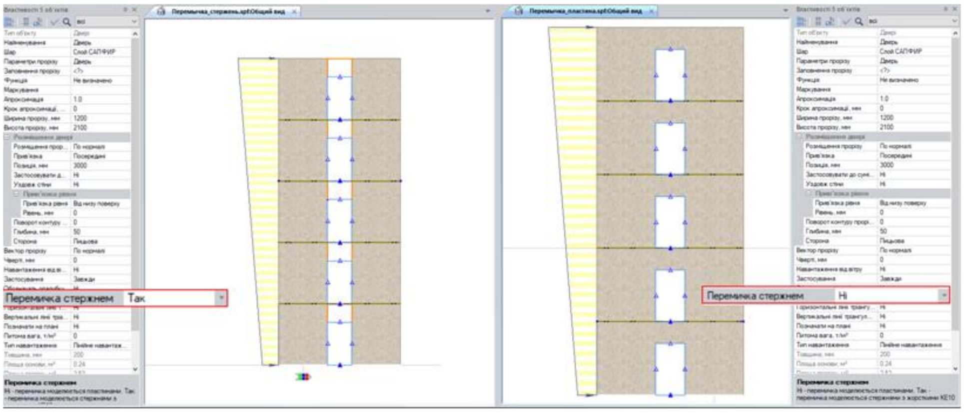

Figure 3.11 – Modelling of slabs

Crack resistance and nonlinearity of concrete. A distinctive feature of reinforced concrete structures is the formation of cracks in the tensile zones of concrete elements. This can be taken into account in the model in several ways. Simplified – use the engineering nonlinearity method as described above (by reducing stiffness). In a fully physically nonlinear analysis, concrete is modelled using special material models with a tensile strength limit: when this limit is reached, the stress no longer increases (or drops significantly) – thus, the element is effectively "cracked" and transmits only insignificant tensile stresses. [19] In LIRA-FEM, a similar effect is implemented through the "constraint on tensile stresses in concrete" option – when the tensile deformation reaches a critical value, the tensile modulus E decreases to almost zero[16]. As a result, the bar in the tensile zone loses its stiffness, and the tensile stress is transferred to the reinforcement (which the model also takes into account due to the plasticity of the reinforcement). For plate elements, a layered model is sometimes used: concrete is divided into layers, which are assigned different moduli – for layers working in tension, the modulus is reduced to simulate cracks. All this significantly complicates the calculation, so engineers often resort to simplifications – for example, they simply take the moment of inertia I_{eff} = 0.5Igross for a cracked cross-section at the operational stage (as recommended by some standards, including DBN). However, in special cases (repeated loading, seismic activity), full-fledged crack modelling may be justified;

- creep and shrinkage. RC structures are subject to time-dependent deformations – creep under constant load and shrinkage during hardening. In calculation models, they can be taken into account either by normative coefficients (increase deformations by ϕ – creep coefficient, reduce E effective: $Eeff = \frac{E}{1+\phi}$) or, more accurately, by stage calculation with long pauses. For example, Eurocode 2 and DBN provide formulas for determining creep deformations after 1000 days, etc. Programs (e.g., MIDAS) have a Time Dependent module for specifying creep: the user enters the creep curve (deformation relative to time under constant stress), and the program integrates it over time, taking into account load hysteresis. [17] LIRA-FEM often uses simplifications: for example, specifying a reduced E modulus according to the duration of the load (for constant loads, $E_{long} = \frac{E}{3}$ according to DBN). It is important that creep causes a redistribution of forces in statically indeterminate systems: stiffer elements (e.g., thick columns) creep less, thinner ones creep more, so over time, the load "flows" to the stiffer elements. The model must take this into account if the task requires an assessment of long-term deflections or forces (for example, to calculate deflection after 5 years of operation). Including creep in the model significantly prolongs the calculation, as many time steps must be simulated, so it is only used when necessary, for example, for prestressed structures or high-rise buildings;

- reinforcement in the model. Depending on the capabilities of the software, reinforcement can be taken into account in different ways. In LIRA-FEM, for bar elements made of reinforced concrete, reinforcement is taken into account through the percentage of reinforcement in the cross-section. In nonlinear calculations, reinforcement is modelled as a separate material within the cross-section with its own stress-strain law (elastic-plastic). In Ansys or Abaqus, reinforcement can be explicitly modelled as separate bars integrated into concrete volume elements (through common nodes or using embedded connections). This approach provides a detailed picture (you can see where the reinforcement is fluid and where it is not), but in large structures it is difficult to introduce tens of thousands of reinforcement elements, so an integral approach is more often used (as in LIRA-FEM).

To sum up: when modelling reinforced concrete, it is necessary to take into account cracks, creep, and the two-component nature of the material (steel + concrete). Simply put, reduce stiffness and use effective modules. More precisely, introduce physical nonlinearity and staging. In any case, after obtaining the results, the engineer must check the limit states according to the standards: for strength – so that the stress in the reinforcement and concrete does not exceed the permissible limits (or so that the reinforcement, if it flows, has the proper plastic reserve); for serviceability – so that deflections and crack widths are within the standards. Some programs (for example, LIRA-FEM) have automatic post-calculation control: according to DBN V.2.6-98:2009, they can calculate the width of cracks based on the forces obtained, or validate the strength of column cross-sections for biaxial bending with compression, etc.

3.14 Features of modelling metal structures

Metal (steel) structures, compared to reinforced concrete structures, have different modelling emphases. Steel is considered a homogeneous material with pronounced elastic-plastic behaviour, and steel structures are more often limited by stability and plasticity rather than cracks (metal does not crack in the engineering sense until destruction). [20] Key points:

- types of connections in the model. In steel frames, it is very important to correctly set the conditions at the joints: whether the beam-to-column joint is hinged (does not transmit torque) or rigid (welded, transmits torque). In the model, this is done either by the type of node connection (some software allows you to choose hinged/rigid) or by inserting a hinge simulation. For example, trusses are modelled as rods with hinged joints, otherwise parasitic moments will arise in the belts and braces, which do not actually exist (in reality, the joint is bolted or hinged). Frame structures of buildings, on the contrary, have rigid joints (welded or bolted with overlays, designed for torque), so they need to be fixed accordingly in the model. Usually, in programs, the default connection of elements is rigid (common joints transmit all 6 degrees of freedom). To make a hinge, the engineer must either split the node and connect them with a hinge (for example, a special "release" element), or specify a connection with free rotation. LIRA-FEM, for example, allows you to set the stiffness coefficients of the node for rotations – 0 means a hinge. Correct configuration of nodes ensures that the model simulates real-life operation: the truss has only axial forces in the braces, the frame has moments in the nodes etc;

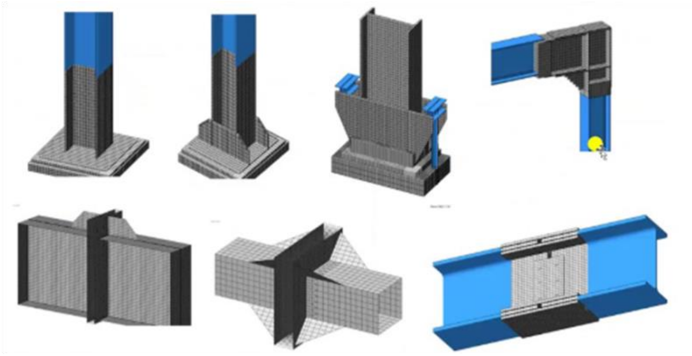

Figure 3.12– Types of connections in metal structures

- eccentricities of elements. In steel structures, elements are often connected not in the centre, but with an offset (for example, a secondary beam is attached to a column at its edge, i.e., the centre of the beam is offset relative to the centre of the column by half the thickness of the column + plate). In models, this is specified by eccentricities: the displacement of the rod line from the connection node by a certain vector. LIRA-FEM has a function for taking into account the eccentricities of beams, which is especially important for diagonal connections, where artificial moments will arise without eccentricity. Eccentricities are also necessary when modelling composite sections – for example, an I-beam with a channel can transfer loads asymmetrically if their centres do not coincide;
- stability and local stability. Unlike reinforced concrete, thin steel elements can lose their load-bearing capacity long before the material undergoes plastic deformation − due to wall buckling, shelf bending, and bending-torsional boiling. These are stability phenomena that do not always manifest themselves in the FEM model (because the element is modelled as a beam with ideal geometry). Therefore, after obtaining the internal forces, it is necessary to perform a limit stability check. Standards [21] (DBN or Eurocode 3) provide formulas for critical stresses for plates, stability coefficients, etc. LIRA-FEM has a "Steel Section Check"

module that automatically calculates the use according to the first group of limit states based on specified standards (e.g., DBN B.2.6-198:2014 or EN 1993-1-1), taking into account overall and local stability. If any of the walls are close to losing stability, the program will issue a warning. However, this is outside the scope of direct calculation – that is, the walls themselves are not modelled as thin elements, and they are checked separately analytically. If necessary, local stability can be modelled explicitly: for example, a model of a plate as a shell can be used to examine its own forms of instability. However, this is rarely done in practice for building structures (except for unusual structures such as round steel shells, where global stability depends on local behaviour);

- physical nonlinearity of steel. Steel is characterized by the phenomenon of fluidity – after exceeding σ_y, it deforms with almost no increase in stress (plastic deformation). In calculations of metal structures under normal loads, it is assumed that stresses should remain in the elastic range (so that there are no residual deformations). However, when checking for progressive failure or seismic activity, plastic redistribution is allowed (the concept of plastic hinges). Modelling the plasticity of steel is similar to that described above: in LIRA-FEM, materials of type "C245-21" represent a bilinear diagram (linear up to σ_y, then horizontal). In nonlinear calculations, the program tracks the achievement of fluidity and then reduces the local shear modulus for the element, allowing it to deform with almost no increase in internal forces. This leads to redistribution: neighbouring elements or sections that are still elastic take on additional loads. As a result, it is possible to identify which elements will become plastic first (which is important for the failure mechanism). If the structure is designed correctly (in the spirit of the "plastic hinge model" concept), plastic hinges occur in less dangerous places (for example, in beams rather than columns – the "column is stronger than beam" principle);

- loss of stability (global). For tall or flexible steel frames, it is essential to take into account second-order effects (geometric nonlinearity). This was discussed in the previous lecture: P–Δ can greatly increase moments. Therefore, the calculation is performed either using a two-step scheme (as recommended by DBN: first linearly, determine the flexibility parameter, then multiply the moments by the coefficient φ), or – better – immediately using nonlinear analysis with geometric nonlinearity. Programs allow this: in LIRA, there is an option to “take into account physical and geometric nonlinearity.” In this case, the calculation will determine whether the structure reaches the critical load. If you need to find the critical coefficient, use linear stability analysis (Euler buckling): solve the problem for the minimum λ at which $d_{et}|K+\lambda K_g|=0$ (where K_g is the geometric stiffness from the base load). Many complexes (SAP2000, Robot) have a Buckling analysis function, which gives critical shapes and coefficients. But this is also taken into account in regulatory checks: Eurocode 3 introduces length coefficients and uses formulas equivalent to this analysis. In any case, the engineer must ensure that the stresses obtained are < critical and that second-order moments are taken into account;

- specific features of elements: trusses, connections. For trusses, make sure that there are no unnecessary moment connections in the nodes of the model. An ideal truss is a hinged-rod system. Nodes in a digital model are often automatically rigid, so they need to be released. One way to do this is to set “hinge at end” in the properties of the members (for example, in RFEM there are checkboxes release M_x, M_y, M_z at the ends of the elements). Another way is to create a small dummy hinge element. After that, the truss under load will work purely in tension/compression (the desired effect). Connections (struts, tie rods) are usually modelled as members, but their possible structural flexibility should be taken into account (for example, if it is a diagonal made of a corner connected with bolts, it may have some slack). Often, for anti-seismic or wind connections, an inflated E-module or area is introduced so that they are guaranteed to take a large force (simulating tension without sagging). If these are tension members (cables), they only work in tension, and when compressed, their stiffness should be zero; this type of nonlinearity can also be specified in some software (element with one-sided action);

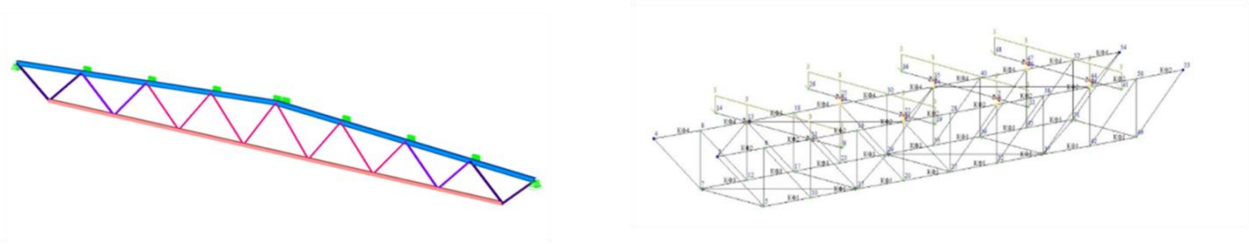

Figure 3.13– Specificity of elements (Source: LIRA-FEM)

- load combinations. For steel structures, it is very important to form design load combinations correctly. This is not so much a question of modelling as it is of post-processing: the results must be evaluated for unfavourable combinations (permanent + temporary + special). Software allows you to automatically generate load combination cases (LCCs) according to normative coefficients (for example, in Robot you can specify EN standards and it will generate all combinations with reliability coefficients and combination coefficients). For modelling, this is indirect: if the structure is nonlinear, then the sequence of load application has an effect. It is usually assumed that all loads increase together (this is what a typical nonlinear analysis does). In reality, for example, permanent loads always act, while wind acts periodically. Several calculations can be performed: one – only permanent + part of variable, another – permanent + other combination. As a result, the worst case is taken;
- assembly stages for steel. In steel structures, assembly is usually quick, there is no creep, so genetic nonlinearity can often be neglected. But there are exceptions: for example, long cantilevered projections are assembled gradually and may require temporary supports. When the temporary stages are removed, the structure redistributes the forces. Such cases are also modelled in stages to see the installation forces (especially important for installation connections).

Therefore, when modelling steel systems, the main focus is on the correct representation of nodes and stability effects. The material nonlinearity of steel (plasticity) can be taken into account when checking for exhaustion of bearing capacity (plasticity reserve), but in operational calculations it is customary to remain in the elastic region. Nevertheless, in modern design based on the limit state concept, if plastic deformations are localized and do not affect performance, they are allowed (the so-called “strong column – weak beam” principle in seismology: beams can flow, absorbing energy, as long as the columns are intact). To analyze such scenarios, a nonlinear calculation with physical nonlinearity is required.

4 MODELING AND CALCULATION OF HIGHLY COMPLEX STRUCTURES

4.1 Modelling of building structures (direct problems, stress control, optimization)

Modern modelling of building structures is based on solving direct problems of structural mechanics. e., determining the stress-strain state (SSS) of elements and systems for given characteristics of the structure, materials, and loads. The direct problem consists in finding the reactions of supports, internal forces, displacements, and deformations of the structure under the action of applied influences using static and dynamic calculation methods. In computer models, this boils down to forming systems of equilibrium equations, which for discretized (finite element) models are written in matrix form:

$$[K]\{u\} = \{F\},$$

where K – the stiffness matrix of the structure, u is the vector of unknown node displacements; F – the vector of external loads[19].

Solving these equations provides a complete picture of the structural stress state: the distribution of forces, stresses, and deflections in the elements.

In addition to traditional direct calculation, stress-strain state control tasks are also relevant, where engineering measures are used to modify the structure or load in order to ensure the desired distribution of stresses and strains. [22] The main idea is to transform stress fields: the initial stress field from the given influences is transformed into a new one that meets the necessary criteria for reliability and efficiency. Such a transformation is achieved by creating additional “control” stresses (corrective fields) that are superimposed on the main stress-strain state and change it through field interference.

Active and passive control. If corrective actions are introduced once – at the stage of manufacturing a new element or reinforcing an existing one – and do not change in the future, this is passive control. Examples include pre-stressing concrete or installing fixed-power vibration dampers. Passive systems operate without an external power supply and are highly reliable, but cannot adapt to changing external conditions[23]. On the other hand, if the structure is equipped with mechanisms that change their influence in real time with changing loads (sensors + actuators), we have active control of the SSC. Active systems allow for more effective vibration damping or redistribution of forces under different load scenarios, but require a power source and complex automation, which increases the cost and may raise questions about reliability[20]. There are also semi-active and hybrid systems that combine passive reliability with the ability to partially adjust parameters (for example, variable dampers with magnetorheological fluid).

Classification of methods for regulating stress. Stress control can be carried out in various ways, which are conventionally divided into the following groups:

1. Changing the characteristics of the structure: changing the geometric configuration of the structure (for example, changing the location of elements, the shape of the structure), compensating for deformations (counter-deformation of elements against expected deflections) and introducing pre-stressing in the elements. Thus, even at the stage of installation or manufacture, the structure is deliberately deformed or stressed in order to obtain a more favourable stress state under working load.

2. Manipulation of trusses and supports: disconnection or rupture of connections in the structure (e.g., cutting a non-separable system into simple beams to remove secondary stresses), adding new connections or supports (adding additional supports under uniform elements, installing tie rods, etc.) or changing the nature of the connections (e.g., hinged connection instead of rigid). Such measures affect the distribution of internal forces by changing the design scheme.

3. Adjustment of design parameters: changing its stiffness characteristics (modernizing cross-sections, adding stiffening ribs, replacing material with stronger

material), inertial parameters (massive counterweights to influence dynamics), dissipative properties (installing dampers to increase vibration damping) or physical and mechanical properties of the material (for example, after the concrete has hardened, it is heated to remove residual stresses etc.).

4. Load control: active disturbance suppression – introduction of additional loads in antiphase to the main ones (as implemented in active vibration dampers that generate a force opposite in phase to the seismic one); partial unloading of the structure (for example, temporary jacking of beams during peak loads); Compensation of loads in another direction (preliminary compression of the structure with a vertical force to increase stability under the action of a horizontal force); filtering of influences – installation of devices that do not allow certain load components to pass (for example, seismic isolators "cut off" high-frequency vibrations that are dangerous for the structure, but allow slow movements to pass).

The methods listed above can be applied both once (passively) and in real time (actively). In many cases, it is rational to combine several approaches—for example, pre-stressing a concrete beam (a passive measure) in combination with an active vibration damper on its supports for seismic safety. As researchers aptly put it, structural control not only improves technical and economic performance, but also significantly increases reliability, with the greatest effect achieved when regulating at all stages of the "life cycle" of an element or system.

Examples of passive control: steel beams with tensioning. One of the classic examples of stress redistribution in a structure is steel beams with tie rods (tension rods). This engineering solution, which was common in historical structures, is essentially an inverted truss: a steel rod (tie rod) fixed to supports runs along the span of the beam. Under load, the beam deflects, and the tie rod absorbs uniform forces, acting as the lower chord of an imaginary truss, while the beam itself mainly bears compression (upper chord). The presence of a tie rod sharply reduces the bending moments in the beam, redistributing them to the axial force in the tie rod, which makes it possible to use a thinner beam and reduce material consumption. Some old designs even had nuts or turnbuckles to adjust the tension of these ties, which let you tighten the beam to make up for sagging from long-term use and minimize deflection. A steel beam with a tie rod is an example of a passive stress control system, where a structural measure (adding a tie rod) changes the operating pattern of the element once and then continues to work without intervention. A similar principle is used in wooden beams in historic buildings (the so-called queen-post truss): two wooden beams as a compressed element and a lower steel tie rod as a tensioned element form a triangular system that effectively spans large spans.

Examples of passive control: prestressed reinforced concrete elements. Another widely used method of controlling stress strain state stress-strain state is the prestressing of reinforced concrete structures. [24] Its essence is that compressive stresses are introduced into the concrete in advance, which counteract the tensile stresses from working loads. This is achieved either by pre-tensioning the reinforcement (which is stretched with jacks before concreting and released after the concrete has hardened, causing the reinforcement to compress the concrete) or by tensioning the concrete (reinforcement ropes are pulled through the hardened concrete and compressed with anchors). As a result, the reinforced concrete beam in operation already has initial compressive stresses that partially or completely compensate for the uniform working stresses from the load [18]. Simply put, a “negative” bending moment or longitudinal force is introduced into the structure, which balances the “positive” moment from its own weight and payload. With a sufficient level of pre-compression, concrete may not experience any tension under standard loads, which prevents the formation of cracks and allows for fuller use of the material's compressive strength. Figure 4.1 schematically shows the difference in the performance of a conventional and a pre-stressed beam: in an unstressed concrete beam, the lower fibres are subjected to tension and cracking, while in a prestressed beam, the initial compressive force in the lower fibres compensates for the tension, and the stresses are distributed more evenly.

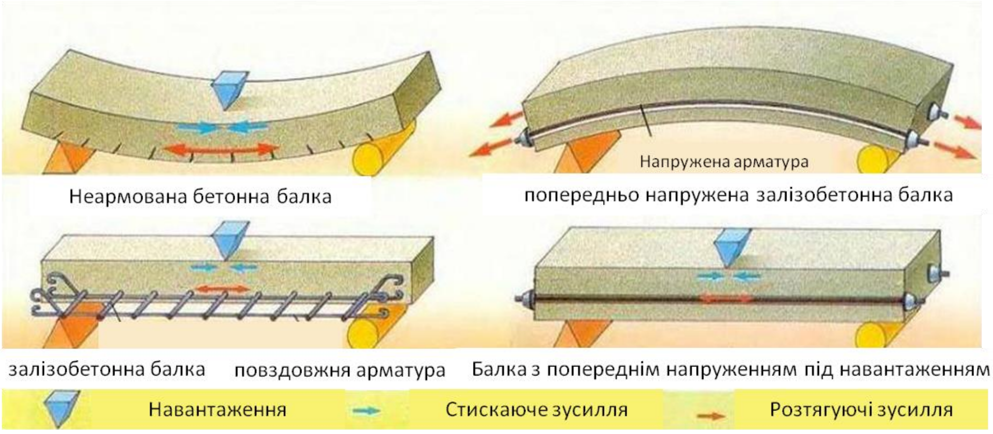

Figure 4.1 – Comparison of stresses in a concrete beam without reinforcement (left) and a prestressed reinforced concrete beam (right)

Preliminary compression of the lower zone of the concrete beam leads to a reduction in tensile stresses under load; the stress diagram becomes balanced.

Prestressing is typically a passive method (no intervention is required after the reinforcement has been tensioned). It significantly increases the strength and stiffness of elements, allowing larger spans to be covered with less deflection. Pre-stressed structures are widely used in bridge construction, in the floors of large-span buildings, in foundation slabs, etc. Specialized software modules are used to calculate them (for example, LIRA-FEM has a mode for specifying the force factors of pre-stressing in elements).

Active stress control systems. In contrast to the passive examples given above, active systems involve dynamic changes in the impact on the structure. An example is an active vibration damper in a building: a heavy mass (pendulum) on a spring suspension with servo drives, installed on the upper floors of a skyscraper. When the building oscillates due to wind or an earthquake, sensors detect the acceleration, and the controller causes the pendulum to move in antiphase to the building's oscillations, damping them. Such a pendulum does not simply passively dissipate energy, but introduces a controlled force that varies depending on the oscillation mode of the structure [23]. Other examples of active control include adaptive supports (jacks capable of changing the height or stiffness of the support during operation) or active prestressing: the structure is equipped with a system of hydraulic jacks that can gradually tighten or loosen the tension of cables or frame elements in response to changes in load [3]. Although such solutions are currently rare due to their complexity, studies show that active control can radically increase the load-bearing capacity and durability of structures, especially in extreme conditions. Hybrid systems are promising: a combination of active actuators with conventional structural elements that provide backup in case of control failure.

Mathematically, strength conditions can be written as a system of equations or an optimization criterion, for example:

$$A - B = 0,$$

where $A_{ij}(x)$ – a matrix of actual (controlled) values of deformation or force factors in the i-th cross section (point) for the j-th parameter; x – vector of control parameters (e.g., prestress values); B_{ij} – matrix of permissible (reference) values of these factors[13]. The goal of control is to achieve equality between actual and reference values (not to exceed the limit deformations and stresses). [13] This forms an optimization problem for the extremum of a certain functional (minimum maximum stress, minimum weight etc.):

$$inf_x a_{ij} x \quad or \quad sup_x a_{ij} x,$$

Where parameters x are selected that optimize the selected objective function a_{ij} (for example, minimize the difference in stresses across the cross-section or the maximum sag of the structure).

Practical examples of stress-strain state regulation. In real-world design, engineers often use passive control methods to improve the efficiency of structures. In addition to the aforementioned prestressing, examples include: beams with camouflaged reinforcement (where the reinforcement is positioned to redistribute stress and prevent cracking in hazardous areas), actively adaptive facades (external elements of a building that change their stiffness depending on wind pressure), pre-curved element systems (for example, arches assembled with initial eccentricity that straighten under load, thus absorbing part of the energy). Another important area is seismic isolators and dampers: although they are passive, they can be considered a control measure – they change the path of seismic loads transmitted to the building, "filtering out" dangerous components. In the latest high-rise buildings, tuned mass dampers (absorbing pendulum masses) are often installed – formally, they are passive (the pendulum oscillates spontaneously), but they are tuned to the building's natural frequencies for effective damping, thus acting as a means of regulating the structure's vibrations. [16]

Parameterization and parametric optimization. Modern information technologies for calculation allow the implementation of parametric models of structures. Parameterization is a way to describe the geometry, load, or properties of a structure through a set of parameters that can be easily changed. For example, a beam may have the following parameters: span L, cross-section height h, modulus of elasticity E. [25] By setting them as variables, an engineer can quickly obtain different model options by varying the parameters. Parametric optimization involves the automatic selection of such parameters to achieve the optimal result (minimum weight, minimum cost, maximum stiffness, etc.). [22] Unlike conventional optimization, where specific values of variables are sought, parametric optimization often explores patterns – for example, optimal cross-sectional dimensions as a function of applied load or span length. This is useful when there is uncertainty in the input data: the engineer obtains not one “static” optimal design, but a whole series of design solutions for different scenarios.

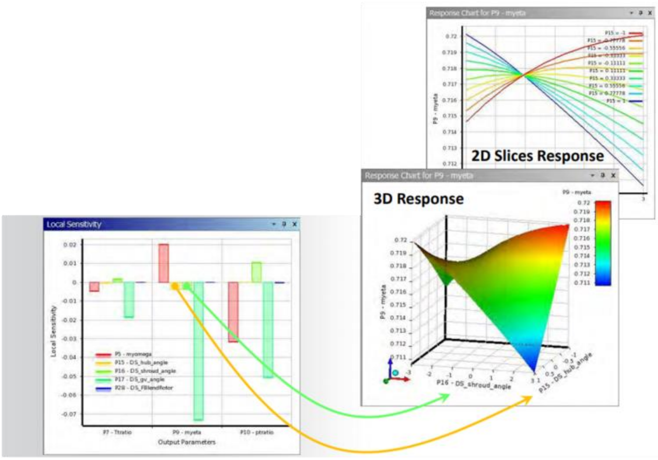

Figure 4.2 – Parametric optimization

Mathematically, the problem can be formulated as follows:

1. Objective function: minimize weight $W = pg\,(A \times L)$ (where p is the density of steel, A is the cross-sectional area, and L is the length of the beam).

2. Constraints: stress $\sigma_{max}(b,h,t_w,t^f)\leq\sigma_{dop}$ (strength not less than permissible), deflection $f(b,h,t_w,t_f)\leq f_{dop}$ (sufficient stiffness), cross-section proportions comply with standards (design constraints).

Optimization methods (gradient, genetic algorithms, optimization improvement method, etc.) change the parameters b, h, t_w, t_f and analyze the model (calculating σ_{max}, σ_{max}, f) until a combination with the smallest W that satisfies all constraints is found. This process can be performed automatically in modern programs (e.g., using scripting languages or built-in optimizers). Parametric optimization also allows dependencies to be taken into account: for example, set the load P as a parameter and find the optimal beam dimensions as a function of P (i.e., solve the problem in general terms for the entire range of loads). In scientific literature, special algorithms of multiparametric optimization are used for this purpose [22]. In practice, however, a series of calculations are often performed with a change in one parameter ("what-if" analysis) and graphs are constructed that show how, for example, the height of the cross-section affects the mass and deflection – this gives the engineer an understanding of the most sensitive parameters of the structure. Topological optimization. A separate modern direction is topological optimization, which seeks the optimal distribution of material within a given area. Unlike parametric optimization, where the structure (topology) of the design is fixed, here the algorithm can add or remove material at any point in the design area. In practice, this is implemented as a gradual "cutting" of material where it has the least impact on the stiffness of the structure. The objective function is usually to minimize elastic energy (or compliance) for a given volume of material. The mathematical formulation is as follows:

Minimize $c(p)=U^TKU$, subject to the constraints: $K(p)\,U = F$,

$$V(p) = \int_{\Omega} p(r)d\,\Omega \leq V_{max}, \quad 0 \leq p(r) \leq 1\,,$$

where p(r) – the material density function (from 0 to 1 at points in the region Ω);c(p) – the compliance (work of external forces, equivalent to elastic deformation energy); K(p) – the stiffness matrix, which depends on the material distribution;U and F – the displacement and load vectors, respectively; V(p) – the volume of material, limited by the specified value V_{max} [26].

Thus we are looking for the distribution p(r) at which maximum stiffness is achieved with the minimum required volume of material.

Topological optimization algorithms are usually iterative methods (SIMP method, ESO/BESO type methods, level set method etc.[26]). During the calculation process, the program gradually reduces the density in areas where the material is "unnecessary" (has little participation in load perception). The result is sometimes unexpected geometries – with curved, graceful shapes similar to natural structures. This is the optimal topology of the structure.

Example. Suppose you need to design a bracket of specified dimensions that is fixed in one area and loaded at a specific point. Topological optimization will "filter out" the material inside the bracket, leaving only the ribs and struts along the paths where the force flows from the load to the support. This creates a shape similar to a branch or bone, without excess material. Figure 4.3 shows a typical result of topological optimization: a solid rectangle is transformed into an optimized structure that is 50 % lighter but has virtually the same stiffness. [27]

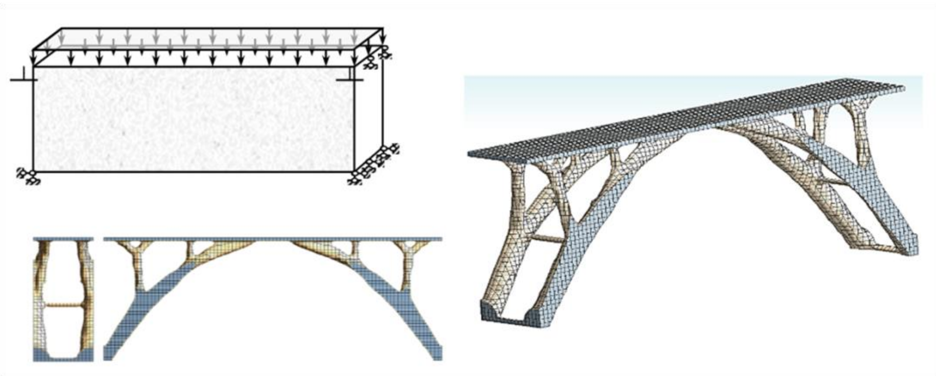

Figure 4.3 – Illustration of topological optimization

The left Figure shows the initial design area (rectangle), fixed on the left and loaded with force from the top right. The right Figure shows the optimal topology of the material: the algorithm left an "intertwined" structure of ribs, which minimizes weight while maintaining load-bearing capacity (source: Number Analytics, 2025).

Advantages and limitations of optimization methods. Optimization (parametric or topological) allows for designs with minimal material consumption, which is especially important in mass production or unique projects (bridges, high-rise buildings) [26]. Optimized shapes often have increased reliability – since stresses are distributed more evenly, there are no obvious areas of overload. At the same time, technological limitations must be taken into account: the shapes produced by the topological algorithm can be difficult to manufacture using traditional methods [26] (although with the development of 3D printing, this problem is partially solved). Optimization also requires significant computing resources and competent input data. The "mathematically best" design does not always take all aspects into account – for example, it is possible to obtain very thin elements that are optimal in terms of rigidity but vulnerable to local loss of stability. Therefore, the optimization result must always be critically analyzed by an engineer and, if necessary, adjusted (for example, adding a safety margin, smoothing out very complex shapes). Despite these caveats, optimization approaches are becoming increasingly common in design practice: there are plugins for BIM systems that perform parametric studies, and packages such as ANSYS, ABAQUS, and Altair OptiStruct have built-in topological optimization modules that engineers can use during the conceptual design stage. [28]

Implementation in modern software complexes. The tasks of managing stress and optimizing structures require appropriate software tools. Most commonly used finite element analysis systems (FEA systems) support tools for such tasks. In particular, the LIRA-FEM program provides a "structure management" mode, where the user can simulate the unloading or reinforcement of elements and analyze the change in forces. For example, you can specify two calculations: one for a conventional frame and the other for the same frame with additional tie rods, and then compare the stresses. LIRA-FEM also has a module for selecting cross-sections according to optimal criteria (minimum weight while complying with strength standards). Another popular package, ANSYS, allows you to perform parametric studies using APDL scripts or the Workbench interface: the engineer sets the model parameters and specifies the objective function (for example, to minimize stress in a part) – ANSYS will automatically change the parameters and perform a series of calculations. For topological optimization, ANSYS offers the Topology Optimization tool, where the user specifies the percentage of mass that can be removed and obtains an optimized shape. Similar capabilities are available in Abaqus / CAE, NASTRAN / Patran, SolidWorks Simulation, and other systems. In recent years, optimization has evolved from an academic concept to a routine tool for practicing engineers.

4.2 Modelling complex structures (high-rise buildings, foundations, dynamics, software)

Modern construction projects increasingly go beyond "typical" conditions– there are high-rise buildings, structures with complex spatial forms, objects on complex engineering and geological foundations, and structures that experience significant dynamic influences (seismic activity, explosions etc.). The calculation of such systems requires a special approach: classical simplified models may be insufficient to take all factors into account. A comprehensive analysis is required using modern computer methods that take into account: (1) the interaction of the building with the ground base, (2) the nonlinearity of materials and structural elements, (3) the influence of dynamics and vibrations. In this lecture, we will consider the features of modelling the most complex cases – from skyscrapers to seismically hazardous structures – and the capabilities of the LIRA-FEM, LIRA 10, RFEM 6, and ANSYS software packages in solving them.

4.2.1 High-rise buildings: loads and stability systems

Specific loads on high-rise buildings. Unlike low-rise structures, where vertical loads play a major role, horizontal loads – wind and seismic forces – are decisive for high-rise buildings. Wind loads increase with height (wind pressure increases, and the area affected by the wind accumulates), so the total bending moment at the base of the building is approximately proportional to the square of the height. This means that at heights above ~100 m, wind is often the dominant factor in frame design. Seismic loads in tall buildings mainly excite low natural vibration modes – fundamental modes, where the entire mass of the building vibrates. Therefore, to ensure seismic resistance, it is necessary to control the deflections of the upper floors and the relative displacements between floors (drift) to avoid excessive deformations and loss of stability. [29]

A tall building must have sufficient spatial rigidity to limit horizontal movements of the top under the influence of wind – this is important for both safety and comfort (excessive swaying causes discomfort to residents). Regulatory documents establish maximum permissible movements of the top, e.g., L/500 or a natural vibration period not less than a certain value, in order to avoid resonance with wind gusts. In addition to overall stiffness, secondary effects such as the P-Δ effect (when a building deflected under load receives an additional moment from the product of weight P and lateral displacement Δ) must be taken into account, as this can cause additional loads in the structure. For tall structures, P-Δ effects can be significant and must be taken into account in the calculation, especially near limit states.

To withstand significant horizontal loads, special bracing systems are used in high-rise buildings. The main types of frame systems for tall buildings include:

- bracing core. This is a central rigid wall structure (usually a reinforced concrete core of the elevator and stairwell). The core, being a closed box, absorbs the lion's share of bending moments from the wind and provides the initial spatial rigidity of the building. However, the core alone may not be sufficient for very tall buildings;

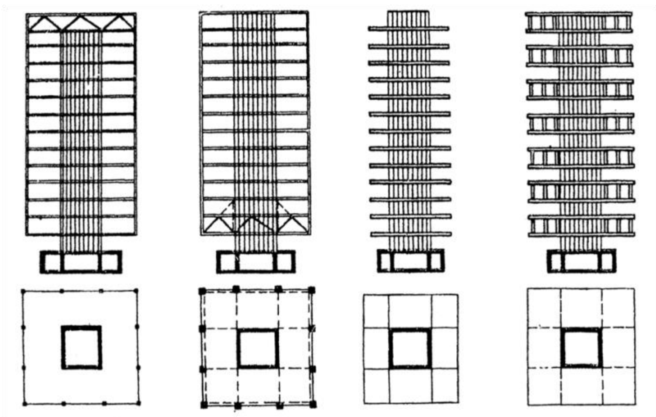

Figure 4.4– Options for the location of the stiffening core

- frame-core system. A combination of a rigid core with a spatial frame structure (columns + rigid beams). The frame works in conjunction with the core, distributing the load more evenly between the centre and perimeter of the building;
- outrigger systems. An outrigger is a horizontal rigid truss or floor slab that connects the core to the columns at the perimeter at a certain height. Outriggers force the outer columns to work in bending, forming a kind of “stiffness triangle” with the core. Thanks to the outriggers, when the top of the building deflects, the peripheral columns on the leeward side are stretched, and those on the windward side are compressed, creating a counteracting moment (similar to a lever). Thus, deflections are reduced and stiffness is increased. Studies show that 2–3 levels of outriggers can reduce deflections by 30–50%. Correct positioning is important: the optimal heights for outriggers are approximately 1/3 and 2/3 of the building height[29]. It is also necessary to take into account the wedging forces in the outriggers from uneven settlement and temperature deformations, so sometimes they are connected with a delay (after the upper floors have been built) or adjustable connections are made to relieve excess stress[29];

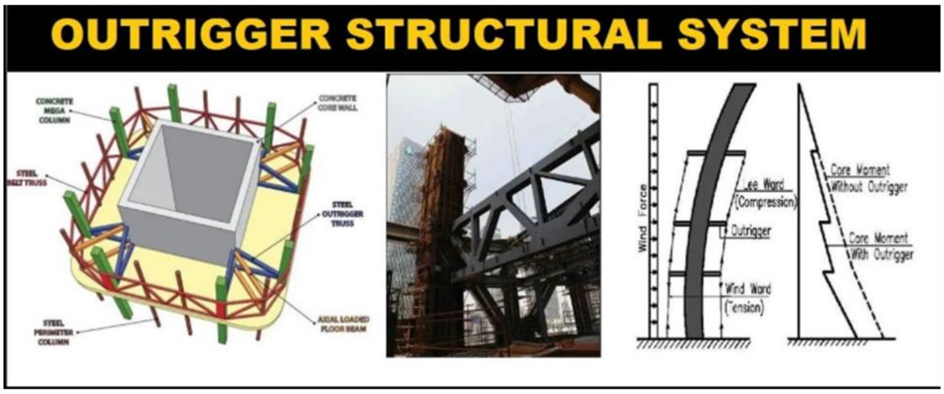

Figure 4.5– Outrigger system

- tubular systems. The concept of the “tube” was revolutionary in the 1960s (engineer Fazlur Khan). The building is designed as a hollow vertical tube: around the perimeter, there are partially spaced columns rigidly connected by beams (tube walls) that take on all the wind loads [30]. Inside, there can be a conventional frame only for vertical loads. Variations: spatial frame tube, braced tube (example – John Hancock Centre with distinctive diagonals on the facade [30]), bundled tube (such as Sears Tower – several tubes joined together for greater rigidity [30]). Tubular systems are extremely effective for very tall structures: the material is concentrated on the outer contours, where the stresses are greatest, and is therefore used rationally. The disadvantage is that a dense grid of columns on the facade can interfere with the architecture, but modern modifications (diagrid, megaframes) partially solve this problem;

- aerodynamic Shape Optimization. This method is more architectural than structural: the building shape is modified to reduce wind loads. Examples include torsional forms (twisting along the height, as in the “Tower of Winds”), stepped façades (to disrupt vortex shedding), rounded corners, and other similar solutions – all of which can decrease the intensity of vortex formation and, consequently, the dynamic wind pressure. Such measures make it possible to reduce design loads or avoid resonant vibrations. Therefore, the architect and the engineer collaborate to geometrically “control” the loading – this is essentially also a way of managing the stress–strain state, but at the level of external influences.

4.2.2 Interaction of the Structure with the Foundation (SSI – Soil-Structure Interaction)

For ordinary buildings, an absolutely rigid foundation model is often used, where the foundation is assumed to be immovable. This approach is acceptable for simple cases, but for tall structures or soft soils it can lead to significant errors, since the soil and the structure act as a single system with mutual deformation influence.

Modelling of the foundation can be performed in various ways. The simplest option is a rigid foundation, where displacements are assumed to be zero. The Winkler elastic foundation model represents the soil as independent springs that respond to pressure proportionally to settlement but does not account for interaction between the springs. The Pasternak model improves upon Winkler’s approach by adding shear stiffness and the mutual influence of settlement zones. A volumetric elastic foundation models the soil in three dimensions with specified elasticity parameters, layering, and other characteristics, which makes it possible to account for stress distribution and nonuniform deformations. A nonlinear foundation model additionally considers plastic deformations, subsidence, separation, or sliding, using contact elements and nonlinear relationships such as Mohr – Coulomb or Drucker–Prager. [31]

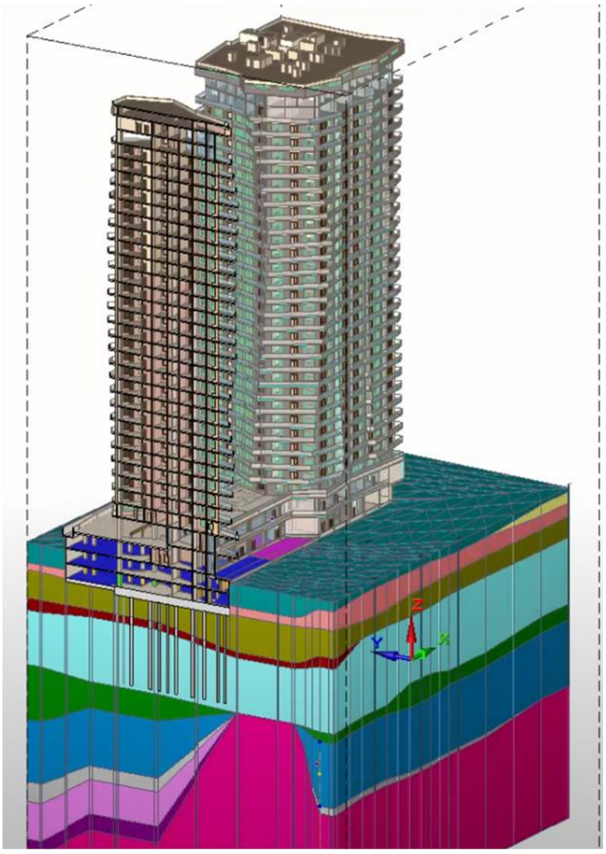

Figure 4.6 – Modelling the Interaction of a Structure with the Foundation

In engineering programs, the "structure–foundation" interaction is implemented through surface elastic elements, volumetric finite elements of soil, or contact interaction. [32] Accounting for the flexibility of the foundation (SSI) can alter the calculation results: the natural vibration period increases, peak forces decrease, but displacements grow. Therefore, it is recommended to create models both with a rigid foundation and with an elastic one.

For soil modelling, it is necessary to specify its physical and mechanical parameters: deformation modulus, Poisson's ratio, unit weight, and for nonlinear models – the internal friction angle, cohesion, and damping. [33]

4.3 Dynamic Loads: Seismic, Harmonic, Impact

4.3.1 Seismic Load

An earthquake acts on a building in the form of inertial forces caused by the accelerations of the ground on which it stands. Essentially, during a seismic shock, the foundation moves rapidly, while the upper part of the building "lags" due to inertia, creating forces $F = m \cdot a$ (where m is the mass of the building elements and *a* is the ground acceleration). The methods of accounting for seismic effects in calculations can be divided into the following categories:

1. Response Spectrum Method**.** This is the most widely used code-based approach. For a given seismic hazard, a response spectrum is constructed (or taken from the design code) – a dependence of the ordinate (acceleration) on the vibration period. The building is modelled in software, its natural modes and periods of vibration are determined, and for each period the acceleration value A(T) is taken from the spectrum. The equivalent force is then calculated as $F = m \times A$. As a result, a set of static forces distributed along the height is obtained, which is applied to the structure. The analysis is performed, and the calculated internal forces are combined for different modes. This method is implemented in most software packages (LIRA-FEM, RFEM) and supports code-based spectra (for example, DBN V.1.1-12:2014 for Ukraine, Eurocode 8, IBC). It allows multiple vibration modes to be considered (the more modes included, the more accurate the distribution of forces along the height). Limitation: the method assumes the structure is elastic and linear; it does not provide information about the time history of motion.

2. Equivalent Static Load Method. This is a simplified approach in which the entire seismic effect is reduced to a horizontal force F_{eq}applied at the top of the building, with its magnitude determined by the formula $F_{eq} = S \cdot m$ (where *S* is the seismic coefficient depending on the region, soil type, and building height). It is applied only to regular, low-rise buildings (for example, up to 5–9 stories), where higher modes are insignificant. In modern software, this method is implemented as a

special case of the response spectrum method (considering only the first mode and the code-specified coefficient). [34]

3. Time History Analysis. This method involves direct integration of the equations of motion of the structure under a given earthquake accelerogram. The equation of motion has the form:

$$[M]\{\ddot{u}(t)\} + [C]\{\dot{u}(t)\} + [K]\{u(t)\} = -[M]\{\Gamma\}a_g(t),$$

where $a_g(t)$ – the ground acceleration record (time-dependent); Γ – the participation vector (usually unity in the degrees of freedom corresponding to the horizontal motion of the foundation).

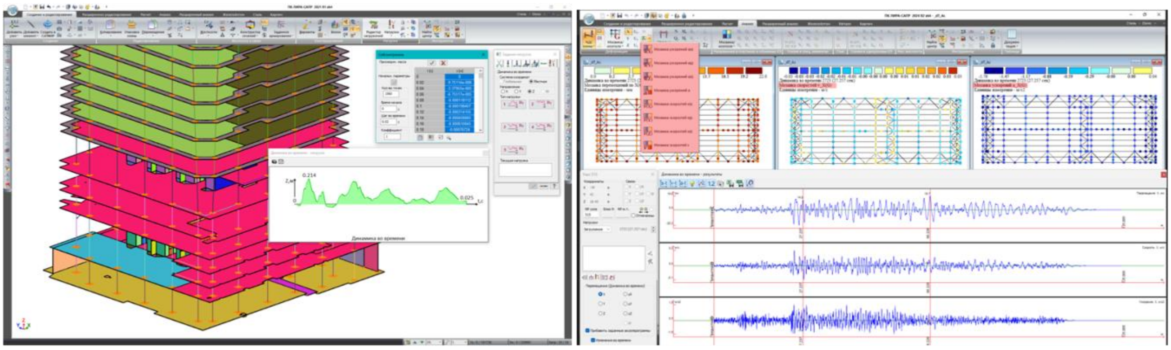

Figure 4.7 – Seismic Loads

The program calculates the response u(t), $\dot{u}(t)$, $\ddot{u}(t)$step by step (with a time step of about 0,01 s) using numerical integration methods (such as the Newmark method, central difference method, etc. [34]). The result is a complete picture of the oscillations: time histories of displacements, velocities, accelerations, and internal forces in the elements. This method is the most accurate and is required for irregular or unique structures (for example, high-rise buildings with pronounced asymmetry). [16] The drawback is that it requires a reliable accelerogram and significant computational resources. LIRA-FEM includes the “Dynamics+” module, which allows performing direct integration (e.g., by the Newmark method) and specifying custom or code-based accelerograms. RFEM 6 also supports accelerogram import and time history analysis. ANSYS and other general-purpose packages can also perform this analysis (in ANSYS, for example, through scripting and the use of C and M matrices for integration). It should be noted that time history results must be processed statistically (for example, by taking envelopes of maximum values or performing spectral decomposition). [35]

Nonlinear dynamic analysis (with stepwise increase, push-over). Although push-over is formally classified as a static method, it is essentially a dynamically equivalent analysis: the equivalent horizontal load is gradually increased on the model until the structure reaches plastic deformation and failure. This method is widely used in the Performance Based Design concept to assess the seismic resistance of high-rise buildings. It provides a capacity curve and shows the plasticity reserve of a building. Programs (LIRA-FEM, SAP2000, ETABS) can perform push-over, but nonlinear hinges must be specified in the elements. For high-rise buildings, push-over is difficult to interpret because they have many forms of vibration, but the method is useful as an additional indicator of strength.

4.3.2 Impact loads and explosions.

An impact is a short-term impulse, such as a heavy load falling or a nearby explosion. In the model, it is defined as a dynamic impulse: either in the form of a force that rises and falls sharply (e.g., triangular impulse, rectangular pressure impulse), or in the form of an equivalent dynamic coefficient. Often, for approximate estimates, the dynamic coefficient β method is used: it is assumed that the impulse lasts for a very short time, and a factor β (e.g., 2 or 3) is added to the static load to account for the effect of inertia. More precisely, a direct numerical analysis is performed. In LIRA-FEM and RFEM, impact actions can be modelled by explicit integration (as a special case of time history): a load history is specified, for example, an impulse with a duration of 0.1 s, and then the behaviour is tracked. [35] A specific case is an explosive load. It consists of a short pressure peak phase and a subsequent rarefaction phase. For structures that may be subject to explosions (industrial buildings, facilities in case of terrorist threats), there are recommendations (UFC 3-340-02, Eurocode) that provide design impulses depending on the mass of explosives and distance. In software, this is again performed either through integration (explicit

FEA, especially for shock waves – sometimes even using CFD to simulate air waves) or through dynamic coefficients.

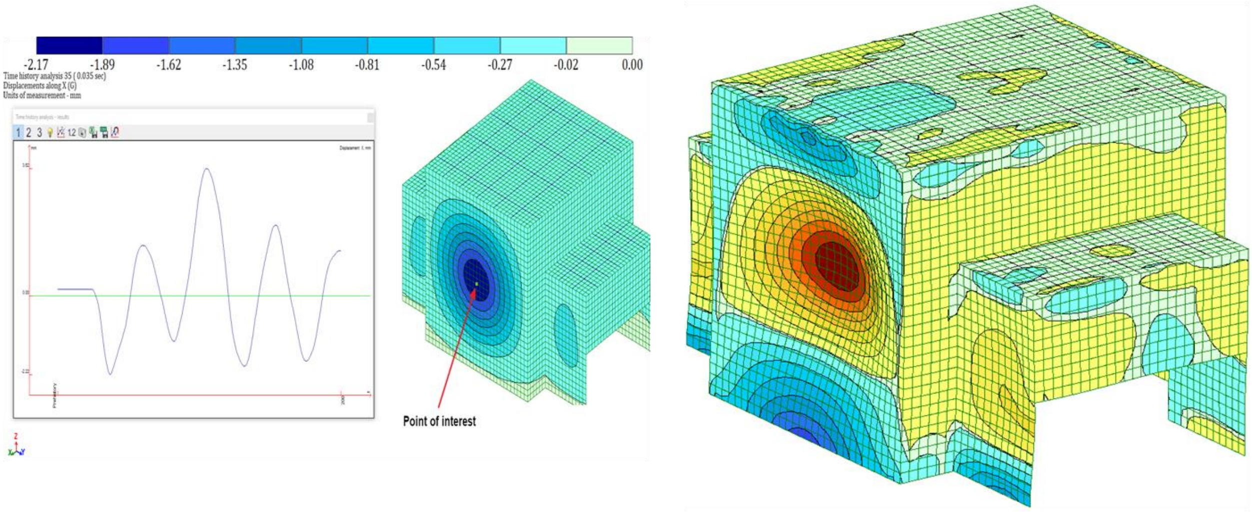

Figure 4.8 – Explosive load

4.3.3 Seismic standards and software implementation

Most countries have building codes for earthquake design (in Ukraine – DBN V.1.1-12:2014). LIRA-FEM, SCAD, SAP2000, ETABS, and other software programs have built-in spectrum databases for various standards. For example, LIRA-FEM supports DBN 2014 and Eurocode 8: the engineer can simply select the intensity (points, soil category) and the program will generate a standard spectrum or equivalent distributed load[31]. RFEM also has a dynamics module where you can import spectra (or set them manually using formulas) and perform a modal calculation. It is very important to normalize the results correctly: the spectral method yields forces that correspond to the linear behaviour of the system with a certain damping, then the standards require the application of behaviour coefficients q (for safety margin due to plasticity). These coefficients are set by the user (this is provided for in LIRA-FEM and RFEM for Eurocode). [36]

In addition, when analyzing tall buildings for seismicity, attention is paid to the forms of vibrations: in very tall structures, the first form is a “pendulum” deflection (the largest contribution), but there may also be higher forms (second, third), which also make a significant contribution to the distribution of forces. It is necessary to ensure that the sum of the involved mass for the considered forms is not less than 90 % of the total mass – then the spectrum calculation is considered accurate.

Regarding interaction with the foundation during an earthquake: as mentioned, it can increase the period and reduce seismic forces, but also increase the settlement and tilt of the structure. The new FEMA P-2091 (2020) handbook emphasizes that the interaction between the foundation and the structure can significantly change its response to an earthquake, so it is important to take it into account for accuracy of assessment[15]. For example, a building on soft soil “rocks” more, but at the same time, part of the energy is dissipated in the soil, meaning less is transferred to the above-ground part.

4.4 Software review: LIRA-FEM, LIRA 10, RFEM6, ANSYS

4.4.1 LIRA-FEM and LIRA10

LIRA is a domestic software package developed back in the 1970s and evolved into the current version of LIRA-FEM. It is focused on building structures and has a user-friendly interface for practicing engineers. It supports the full calculation cycle: from model creation (frame, slabs, shells, solids) to reinforcement selection and verification according to DBN. For complex structures, LIRA-FEM offers:

- physically nonlinear analysis (taking into account material plasticity, crack formation in concrete, and the behaviour of elements after loss of stability);
- taking into account the soil foundation – both in the form of a bedding coefficient and in the form of a volumetric soil model with contacts;
- dynamic calculations – modal analysis, spectral seismic calculation, harmonic analysis, direct integration (option “Dynamics+”);
- special modules: for example, calculation for progressive collapse, assessment of bearing capacity according to a nonlinear concrete diagram, etc.

LIRA 10 (the predecessor of LIRA-FEM) is also widely used, especially in post-Soviet countries. It has a slightly different interface but similar capabilities.

LIRA allows you to simulate a high-rise building with all the factors mentioned above: set the stiffness core (wall elements), outriggers (beam elements with ties), soil model under the foundation (e.g., set the bedding coefficient depending on the layer depth), seismic activity according to DBN, taking into account the necessary vibration forms, wind load (automatically generated according to the specified wind region and height). The engineer has access to the results: vibration modes, period values, load distribution by height, foundation settlement graphs, etc. Importantly, LIRA contains a large block of regulatory checks – for example, after analysis, you can immediately check the strength of columns and walls, the stability of frames, etc. according to DBN. This is convenient for designers. [37]

4.4.2 RFEM6 (Dlubal)

RFEM is a modern 3D FEA complex from Dlubal (Germany). Version 6 is integrated with BIM, has a powerful graphics editor, and modular architecture. For complex structures, RFEM offers:

- modelling of arbitrary geometry. It is particularly useful for geometrically complex shapes (e.g., tensile shells, dihedral facades) because it supports import from Revit, Rhino/Grasshopper, and has an API for custom algorithms;
- add-ons for soil and dynamics. There is a Geotechnical Analysis module that allows you to create foundation models (including nonlinear soil properties). There is a dynamic module RF-DYNAM Pro for modal and seismic analysis. It allows you to import earthquake accelerograms and perform time history, as well as take into account Rayleigh damping. Nonlinear contact interaction (foundation with soil, elements with each other) is also supported through special specified joint nonlinearities;
- BIM integration. RFEM 6 is positioned as part of the BIM environment: the model can be exported/imported in IFC, dxf, Revit link, and other formats. This is important for complex projects where architects, designers, and geotechnical engineers exchange data.

For example, for a skyscraper, you can model the frame and core in RFEM, add a slab on an elastic soil foundation with solid soil elements (using the "Solid" function and "Soil" material type), then apply seismic loading via the Eurocode 8 spectrum (the program has a library of standards) and perform the analysis. The results – deformations, stresses, reactions – can be evaluated both numerically and visually (isolines, 3D view). RFEM also allows nonlinear analysis of concrete elements (e.g., crack formation according to the Diaphragm model). This is useful for evaluating the actual stiffness distribution in core walls during an earthquake.

4.4.3 ANSYS and other universal FEA

ANSYS is a multifunctional finite element analysis complex used not only in construction. For our topics, it is important that ANSYS has:

1. Advanced dynamic analysis capabilities (modal, harmonic, random vibration analysis, time integration, including explicit integration for explosions). For example, the ANSYS Explicit Dynamics module allows you to simulate shock waves, collapses, and impacts using auto-adaptive algorithms and taking into account nonlinear materials.
2. Extensive material libraries, including soils (you can specify soil compression curves, Drucker – Prager models for soils, taking into account plasticity).
3. Optimization capabilities. ANSYS has the Design Explorer tool (for parametric optimization) and Topology Optimization, as mentioned above. This allows you to optimize, for example, the shape of wind connections or the mass of a damper in a high-rise building.
4. The disadvantage for civil engineers is that ANSYS does not "know" building codes and does not have built-in checks for reinforced concrete or steel. This means that after the calculation, the engineer must process the data themselves (e.g., export the forces in the column and calculate the reinforcement using DBN formulas). Therefore, ANSYS is more often used for non-standard tasks – for example, studying seismic response taking into account the elastic-plastic

deformation of elements when it is necessary to see the behaviour beyond the elastic limit (which is difficult to do in LIRA or RFEM due to their regulatory orientation).

When choosing software for modelling complex structures, engineers consider the complexity of the geometry, the need for optimization, regulatory requirements, budget, and the availability of the program. In Ukraine, LIRA-FEM is the standard for structural calculations (it has accumulated experience and complies with DBN). RFEM is gaining popularity thanks to its modern interface and good Eurocode support. ANSYS remains a tool for special tasks and scientific research. In many projects, combined use is justified: for example, the general calculation of a high-rise frame can be done in LIRA to determine forces and select cross-sections, while a separate node or subsystem (say, a foundation slab with a pile field on a complex base) can be analyzed in detail in ANSYS with 3D soil modelling and nonlinear pile-soil contacts. The results are then compared and, if necessary, the design decisions are adjusted.

4.5 The use of artificial intelligence in modern calculations for buildings and structures

Today, the field of design and calculation of building structures is undergoing a significant digital transformation. The introduction of technologies such as Building Information Modelling (BIM), digital twins, and artificial intelligence (AI) is radically changing traditional approaches to design, analysis, and construction. In particular, AI provides powerful new tools that complement classic engineering calculation methods and automate routine processes [38]. The introduction of modern technologies, including AI, makes it possible to increase the productivity of the industry and solve problems that previously required significant time expenditures. It is no coincidence that the topic of applying artificial intelligence in construction is now considered extremely relevant and promising – it is about the need for continuous improvement of technologies and the development of standards for the effective and safe implementation of these innovations. Below, we will look at the new opportunities that AI opens up in structural calculations and what alternative software tools can already help engineers today. [39]

4.5.1 Application of artificial intelligence in structural calculations

Acceleration of numerical analysis. AI algorithms can significantly speed up engineering calculations, in particular using the finite element method, by using surrogate models based on neural networks. This allows analysis results to be obtained in near real time without a full model run, while maintaining acceptable accuracy. After training on classical calculation data, neural network surrogates can instantly predict, for example, maximum bending moments or displacements under various loads.

Optimization of design parameters and shapes. AI is used for automated search for optimal solutions based on criteria of strength, stiffness, and cost-effectiveness. Evolutionary algorithms, deep learning, and generative design make it possible to quickly sift through thousands of options, offering new design configurations that an engineer might not have considered. This approach results in lighter, more stable, and more efficient structures. [40]

Condition monitoring and reliability prediction. AI in Structural Health Monitoring systems automatically detects damage, analyzes sensor and Figure data, predicts remaining life and extreme impacts. This allows for timely repair planning, failure prevention, and structural safety throughout the entire life cycle.

4.5.2 Alternative calculation tools (outside the finite element method

Despite the dominance of complex specialized calculation programs (such as FEA packages – SCAD, LIRA, SAP2000, etc.), engineers widely use simpler tools in their daily practice. These include both universal office tools and specialized programs and web services. Below are the main alternatives that are not directly based on the finite element method but help to perform engineering calculations:

- Spread sheets (Excel). At the preliminary design and verification stage for simple elements, engineers often use Microsoft Excel, which has become the standard due to its accessibility and convenient “reactive” interface. It allows you to quickly implement formulas and immediately see the result. Disadvantages: difficulty in tracking versions, risk of errors in formulas, and reduced readability of large files. Therefore, strict documentation and independent verification are required;
- specialized calculation programs (Mathcad, Tedds). These combine the flexibility of Excel with clarity and standardization. Mathcad allows you to format formulas in mathematical form with comments and graphs, while Tedds offers ready-made calculation templates and automatic report generation. Advantages: transparency, versioning, reusability. Disadvantages: less flexibility for non-standard tasks and the need for training;
- online calculators. Services such as Eurocode Applied, ClearCalcs, and SkyCiv allow you to perform standard calculations through a browser without installing software, automatically take standards into account, and reduce the risk of errors. They are suitable for typical tasks but do not replace powerful modelling tools for complex structures.

4.5.3 Prospects and future trends

Innovative solutions based on artificial intelligence (AI) technologies are already demonstrating a significant transformational impact on the field of engineering calculations and building design. According to industry experts, the widespread introduction of AI will not only automate a significant portion of routine computational operations, but also significantly expand the range of structural solutions considered at the design stage.

The use of AI-assisted design methods makes it possible to generate and evaluate a significantly larger number of alternative design options in a short period of time than is possible with the traditional manual approach. This approach facilitates a more complete exploration of the solution space, the search for optimal configurations, and an improvement in the quality of the final design solutions. [26]

One of the key advantages of implementing AI is its impact on the sustainability and safety of structures. [41] Optimization algorithms make it possible to reduce the amount of materials used and select rational shapes and structural elements, which lowers both construction costs and environmental impact. In addition, the use of AI-based predictive monitoring facilitates the timely detection of potential defects or failures, increasing the reliability of structures and extending their service life.

At the same time, the widespread introduction of AI is accompanied by a number of challenges. These include the need to develop uniform standards and methodologies for integrating AI into the design and operation of structures. Of particular importance are the requirements for transparency and explainability of algorithms, the ability to independently verify the results obtained before their practical application, as well as ensuring the cybersecurity of systems that work with connected sensors and cloud services.

QUESTIONS FOR SELF-CHECK

1. Why is it critically important for a design engineer to be proficient in modern computer-based calculation methods? Give 5–7 specific arguments.

2. Explain how regulatory documents regulate the use of computer models in structural design and how this is related to the assessment of their reliability and safety under limit states.

3. Name the main Eurocode documents and briefly indicate what each of them regulates.

4. How are load combinations formed according to DBN V.1.2-2:2006? Explain the differences between basic, partial, and special combinations.

5. How is the compliance of software packages (LIRA-FEM, RFEM 6,ANSYS/Abaqus) with the requirements of DBN/Eurocodes confirmed? Which settings play a role and what must be checked manually?

6. What types of idealizations are used when creating a model and how do they affect the results of the stress-strain state (SSS)?

7. How to correctly model supports (hinge, clamp, movable support) and how does the choice of fastening type affect the results?

8. In what cases and how are elastic connections (elastic foundation coefficient) used to model the compliance of the soil foundation?

9. Explain the difference between hinged, rigid, and semi-rigid joints. Give examples for metal trusses and monolithic frames (column-beam joint).

10. Describe the main simplified material models and indicate the limits of their application.

11. When is it appropriate to use equivalent stiffnesses (e.g., for composite floors and joints), and how is the acceptability of such simplification verified?

12. Describe what comparisons and control examples should be performed to confirm the adequacy of the model.

13. What are the three main groups of errors? Give examples and ways to identify and reduce them.

14. How is mesh convergence analysis performed and how should singularities (sharp corners, concentrated forces/rigid fixings) be handled?

15. Explain the difference between analytical and numerical approaches; when analytics become unsuitable and why FEM is the default tool in software.

16. Formulate the idea of FEM: discretization, shape functions, nodal unknowns; write down the general form of the system Ku = F and explain what K, u, F mean.

17. How to choose the type/order of elements and mesh size in terms of accuracy and computational cost?

18. How are boundary conditions/loads implemented in an FEM model and what changes in the K matrix and F vector?

19. What are the basic assumptions of linear structural theory and when is the superposition principle correct?

20. How to correctly set up a model in linear analysis (loads, combinations, boundary conditions) for further verification according to standards?

21. What is the physical meaning of linear stability analysis (eigenvalues/modes of instability) and what are the limitations of this approach?

22. Compare the spectral method (eigenforms/frequencies + spectrum) and direct integration of equations of motion (over time). When is each method more appropriate in design practice?

23. What are the typical settings in time integration and the risks of convergence/stability of calculations?

24. List the types of engineering nonlinearity and give engineering examples of each.

25. What are the main methods for solving nonlinear problems and how to track errors?

26. How to model the stages of construction/assembly, creep, and crack formation, taking into account the load history?

27. What are the key features of modelling reinforced concrete and steel, and which software tools are critical here?

28. When is a joint “structure-foundation” model needed, and how does it affect settlement and stress-strain state? What should be considered in the case of seismic and impact/explosive effects?

29. What criteria should be used to select software (LIRA-FEM/LIRA 10, RFEM 6, ANSYS/Abaqus) for a specific task, and how can artificial intelligence be used in calculations and checks?

REFERENCE LIST

1. Dynamic Analysis Software (LIRA-FEM) [Electronic resource]. – Access mode: https://liraland.com. – Date of access: 22.08.2025.
2. FEA Academy. History of the Finite Element Method [Electronic resource]. – Access mode: https://www.fea-academy.com/index.php/component/content/article/27-blog/fea-generalities/73-fem-history?Itemid=101. – Date of access: 22.08.2025.
3. CAE Assistant. Abaqus Nonlinear Analysis vs Linear Analysis (Tutorial 2023) [Electronic resource] // CAEassistant.com. – 2023. – Access mode: https://caeassistant.com/blog/abaqus-nonlinear-analysis-vs-linear-video/. – Date of access: 22.08.2025.
4. Програмний комплекс ЛІРА-FEM. Приклади розрахунку і проєктування: Textbook [Electronic resource]. – Kyiv: LIRA-LAND, 2022. – 635 p. – Access mode: https://liraland.com. – Date of access: 22.08.2025.
5. Murad J. Errors in FEA and Understanding Singularities (Beginners' Guide) [Electronic resource] / J. Murad // SimScale Blog. – Access mode: https://www.simscale.com/blog/errors-in-fea-and-singularities/. – Date of access: 22.08.2025.
6. Novedge. Design Software History: Evolution of Structural Analysis Software – From Manual Calculations to AI-Driven Tools [Electronic resource] / Novedge // Novedge Blog. – 12.07.2024. – Access mode: https://novedge.com/blogs/design-news/design-software-history-evolution-of-structural-analysis-software-from-manual-calculations-to-ai-driven-tools. – Date of access: 22.08.2025.
7. Eastman C., Teicholz P., Sacks R., Liston K. BIM Handbook: A Guide to Building Information Modeling for Owners, Designers, Engineers, Contractors, and Facility Managers [Electronic resource]. – 3rd ed. – Wiley, 2018. – 688 p. – ISBN 978-1-119-28753-7. – Access mode: https://www.researchgate.net/publication/326270076_BIM_Handbook_A_Gui

de_to_Building_Information_Modeling_for_Owners_Designers_Engineers_Contractors_and_Facility_Managers. – Date of access: 22.08.2025.

8. Загальні принципи забезпечення надійності та конструктивної безпеки будівель і споруд : ДБН В.1.2-14:2018 (Державні будівельні норми України). – Чинний від 2016-01-01. – Kyiv: Ministry of Regional Development of Ukraine, 2016. – 56 p. – Access mode: V1214-2018.pdf. – Date of access: 22.08.2025.
9. What is FEM in Structural Analysis? [Electronic resource] // SDC Verifier. – Access mode: https://sdcverifier.com. – Date of access: 22.08.2025.
10. Zienkiewicz O. C., Taylor R. L. The Finite Element Method: Its Basis and Fundamentals [Electronic resource]. – 7th ed. – Oxford: Butterworth-Heinemann (Elsevier), 2013. – 752 p. – Access mode: https://www.sciencedirect.com/book/9781856176330/the-finite-element-method-its-basis-and-fundamentals. – Date of access: 22.08.2025.
11. HSH – STRAUS7. Buckling Analysis – Linear vs Nonlinear [Electronic resource] // HSH Info. – 2001. – Access mode: https://www.hsh.info/articl2.htm. – Date of access: 22.08.2025.
12. Dlubal Software. RFEM 6 – Structural Analysis and Design Software: Online Manual [Electronic resource]. – Tiefenbach: Dlubal Software GmbH, 2022. – Access mode: https://www.dlubal.com/en/downloads-and-information/documents/online-manuals. – Date of access: 22.08.2025.
13. CAE Assistant. Abaqus Nonlinear Analysis vs Linear Analysis (Tutorial 2023) [Electronic resource] // CAEassistant.com. – 2023. – Access mode: https://caeassistant.com/blog/abaqus-nonlinear-analysis-vs-linear-video/. – Date of access: 22.08.2025.
14. Dassault Systèmes. ABAQUS 6.14 Analysis User’s Guide [Electronic resource]. – Providence, RI, USA, 2014. – Access mode: https://classes.engineering.wustl.edu/2009/spring/mase5513/abaqus/docs/v6.6/books/usb/pt03ch06s01aus36.html. – Date of access: 22.08.2025.

15. Городецький О. С., Мішалов Г. М., Лазарєв О. М. Нелінійна будівельна механіка з ПК ЛІРА-САПР. – Kyiv: Lira Service, 2020. – 280 p.
16. A Critical Overview of the Published Research on Control of Vibrations of Civil Engineering Structures [Electronic resource]. – 1999. – Access mode: https://link.springer.com/chapter/10.1007/978-94-011-4611-1_20. – Date of access: 22.08.2025.
17. Eurocode 2: Design of Concrete Structures – Part 1-1: General Rules and Rules for Buildings : EN 1992-1-1:2004 (European Standard). – Effective from 2004-01-01. – Brussels: CEN, 2004. – Access mode: https://www.phd.eng.br/wp-content/uploads/2015/12/en.1992.1.1.2004.pdf. – Date of access: 22.08.2025.
18. Nawy E. G. Prestressed Concrete: A Fundamental Approach [Electronic resource]. – 5th ed. – Upper Saddle River: Prentice Hall, 2009. – 936 p. – ISBN 978-0136081500. – Access mode: https://railtec.illinois.edu/wp/wp-content/uploads/Nawy-2009-Prestressed-Concrete.pdf. – Date of access: 22.08.2025.
19. Бетонні та залізобетонні конструкції. Основні положення : ДБН В.2.6-98:2009 (Державні будівельні норми України). – Чинний від 2011-01-01. – Kyiv: Ministry of Regional Development and Construction of Ukraine, 2011. – 84 p. – Access mode: https://e-construction.gov.ua/laws_detail/3200410998024438840. – Date of access: 22.08.2025.
20. Сталеві конструкції. Норми проектування : ДБН В.2.6-198:2014 (Державні будівельні норми України). – Чинний від 2014-10-01. – Kyiv: Ministry of Regional Development of Ukraine, 2014. – 290 p. – Access mode: https://uscc.ua/uploads/page/Figures/normativnye%20dokumenty/dbn/dbn_v_2_6-198_zmina1_interactive.pdf. – Date of access: 22.08.2025.
21. Eurocode 3: Design of Steel Structures – Part 1-1: General Rules and Rules for Buildings : EN 1993-1-1:2005 (European Standard). – Effective from 2005-01-01. – Brussels: CEN, 2005. – Access mode: https://gaprojekt.com/wp-

content/uploads/2021/11/Eurocode-3-Design-Of-Steel-Structures.pdf. – Date of access: 22.08.2025.
22. Parametric Optimization for Structural Design Problems [Electronic resource]. – 2019. – Access mode: https://www.researchgate.net/publication/337530561_Parametric_Optimization_for_Structural_Design_Problems. – Date of access: 22.08.2025.
23. Casciati F., Rodellar J., Yıldırım U. Active and Semi Active Control of Structures – Theory and Applications: A Review of Recent Advances [Electronic resource] / F. Casciati, J. Rodellar, U. Yıldırım // Journal of Intelligent Material Systems and Structures. – 2012. – Vol. 23, No. 11. – P. 1181–1195. – DOI: 10.1177/1045389X12445029. – Access mode: https://doi.org/10.1177/1045389X12445029. – Date of access: 22.08.2025.
24. Eng-Tips Forums. Historic Mill Building Beam (discussion thread #312160) [Electronic resource]. – 2011. – Access mode: https://www.eng-tips.com/threads/historic-mill-building-beam.312160/. – Date of access: 22.08.2025.
25. Спіранде К. В. Інформаційні технології розрахунку та проєктування будівельних конструкцій. – Kharkiv: Kharkiv National University of Civil Engineering and Architecture, 2021. – 52 p.
26. Topology Optimization in Structural Engineering [Electronic resource]. – Access mode: https://www.numberanalytics.com/blog/topology-optimization-structural-engineering. – Date of access: 22.08.2025.
27. Bendsøe M. P., Sigmund O. Topology Optimization: Theory, Methods, and Applications [Electronic resource]. – Berlin: Springer-Verlag, 2003. – 370 p. – ISBN 978-3540429920. – Access mode: https://www.researchgate.net/publication/261174060_Topology_Optimization_Theory_Method_and_Applications. – Date of access: 22.08.2025.
28. Барабаш М. С., Кір'язєв П. М., Лапенко О. І., Ромашкіна М. А. Основи комп'ютерного моделювання. – Kyiv: Lira-Land, 2018. – 492 p. – ISBN 978-966-932-077-3.

29. The Evolution of Outrigger System in Tall Buildings [Electronic resource]. – Access mode: https://global.ctbuh.org/resources/papers/download/2736-the-evolution-of-outrigger-system-in-tall-buildings.pdf. – Date of access: 22.08.2025.
30. Ali M. M., Al-Kodmany K. Structural Systems for Tall Buildings [Electronic resource] / M. M. Ali, K. Al-Kodmany. – Access mode: https://www.researchgate.net/publication/361813905_Structural_Systems_for_Tall_Buildings. – Date of access: 22.08.2025.
31. FEMA P-2091. A Practical Guide to Soil-Structure Interaction [Electronic resource]. – Washington D.C.: Federal Emergency Management Agency, 2020. – 218 p. – Access mode: https://www.fema.gov/sites/default/files/documents/fema-p-2091-soil-structure-interaction.pdf. – Date of access: 22.08.2025.
32. Hibbeler R. C. Structural Analysis: Textbook (Ukrainian translation of Structural Analysis, 8th ed.). – Kyiv: National Aviation University, 2016. – 928 p. – ISBN 978-966-598-876-2.
33. Колчунов В. І., Сердюк В. М. Взаємодія споруд з ґрунтовою основою: Textbook. – Rivne: Publishing House of National University of Water and Environmental Engineering, 2013. – 248 p. – ISBN 978-966-327-263-0.
34. Barabash M. S., Pisarevskyi B. Y., Bashynskyi Y. Taking into Account Material Damping in Seismic Analysis of Structures [Electronic resource] / M. S. Barabash, B. Y. Pisarevskyi, Y. Bashynskyi // Tehnički Glasnik. – 2020. – Vol. 14, No. 1. – P. 55–59. – DOI: 10.31803/tg20180523192812. – Access mode: https://doaj.org/article/78e85ec4d75b499a9b173c63997083d5. – Date of access: 22.08.2025.
35. Будівництво в сейсмічних районах України : ДБН В.1.1-12:2014 (Державні будівельні норми України). – Чинний від 2014-10-01. – Kyiv: Ministry of Regional Development of Ukraine, 2014. – 110 p. – Access mode: https://zakon.isu.net.ua/sites/default/files/normdocs/dbn_v.1.1-

12_2014_budivnictvo_v_seysmichnikh_rayonakh_ukraini.pdf. – Date of access: 22.08.2025.

36. Eurocode 8: Design of Structures for Earthquake Resistance – Part 1: General Rules, Seismic Actions and Rules for Buildings : EN 1998-1:2004 (European Standard). – Effective from 2004-01-01. – Brussels: CEN, 2004. – 229 p. – Access mode: https://www.phd.eng.br/wp-content/uploads/2015/02/en.1998.1.2004.pdf. – Date of access: 22.08.2025.
37. ЛІРА-САПР. Опис можливостей програмного комплексу [Electronic resource]. – LIRA-SAPR 2021 (online documentation). – Kyiv: LIRA-LAND, 2021. – Access mode: https://www.liraland.com/lira/release-notes/2024/soil. – Date of access: 22.08.2025.
38. AI-powered Optimization for Structural Design [Electronic resource]. – Access mode: https://novaedgeusa.com. – Date of access: 22.08.2025.
39. Azanaw G. M. Revolutionizing Structural Engineering: A Review of Digital Twins, BIM, and AI Applications [Electronic resource] / G. M. Azanaw // Indian Journal of Structure Engineering. – 2024. – Vol. 4, No. 2. – P. 1–8. – DOI: https://doi.org/10.54105/ijse.B1321.04021124. – Date of access: 22.08.2025.
40. The Role of AI in Transforming Structural Engineering [Electronic resource] // Neural Concept. – Access mode: https://neuralconcept.com. – Date of access: 22.08.2025.
41. Донченко М. В. Технології комп’ютерного проєктування: Textbook. – Mykolaiv: Petro Mohyla Black Sea National University, 2021. – 364 p. – ISBN 978-966-336-430-8.

Електронне навчальне видання

РЕЗНІК Петро Аркадійович
ЛУГЧЕНКО Олена Іванівна

КОМП'ЮТЕРНІ МЕТОДИ РОЗРАХУНКУ БУДІВЕЛЬНИХ КОНСТРУКЦІЙ

КОНСПЕКТ ЛЕКЦІЙ

(для здобувачів другого (магістерського) рівня вищої освіти денної та заочної форм навчання зі спеціальності192 – Будівництво та цивільна інженерія, освітня програма «Промислове та цивільне будівництво»)

(Англ. мовою)

Відповідальний за випуск П. М. Фірсов
За авторською редакцією
Комп'ютерне верстання *П. А. Резнік*

План 2025, поз.271Л

Підп. до друку 24.11.2025. Формат 60 × 84/16.
Ум. друк. арк. 6.4.

Видавець і виготовлювач:
Харківський національний університет
міського господарства імені О. М. Бекетова,
вул. Чорноглазівська, 17, Харків, 61002.
Електронна адреса: office@kname.edu.ua
Свідоцтво суб'єкта видавничої справи:
ДК № 8386 від 14.07.2025.