MINISTRY OF EDUCATION AND SCIENCE OF UKRAINE

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

Methodological recommendations

for practical classes and organizing independent

on an academic discipline

«COMPUTER METHODS FOR CALCULATING BUILDING STRUCTURES»

(for students of the second (master's) level of higher education all forms of education speciality 192 – Building and Civil Engineering, of education program "Industrial and civil construction")

**Kharkiv
O. M. Beketov NUUE
2026**

Methodological recommendations for practical classes and organizing independent on an academic discipline «Computer methods for calculating building structures» (for students of the second (master's) level of higher education all forms of education speciality 192 – Building and Civil Engineering, of education program "Industrial and civil construction") / O. M. Beketov National University of Urban Economy in Kharkiv ; comp. : P. A. Reznik, O. I. Lugchenko. – Kharkiv : O. M. Beketov NUUE, 2026. – 93 p.

Compliers: PhD (Tech.), Associate Professor P. A. Reznik,
PhD (Tech.), Associate Professor O. I. Lugchenko

Reviewer PhD (Tech.), Associate Professor of Building Structures Department of O. M. Beketov National University of Urban Economy in Kharkiv Oleg Kalmykov

Recommended by the Department of Building Structures, Meeting Minutes № 8 on 16.01.2025

The methodological recommendations contain a set of practical tasks and exercises for the academic discipline «Computer methods for calculating building structures», aimed at reinforcing the acquired knowledge, skills, and competencies.

CONTENT

INTRODUCTION

The rapid development of information technologies is driving significant changes in engineering practices related to the design and analysis of building structures. Modern construction is inconceivable without the use of computer-based analysis methods, which enhance calculation accuracy, improve the efficiency of structural solutions, and enable the modelling of complex spatial systems.

The academic discipline “Modern Computer Methods for Structural Analysis” is one of the core subjects in the training of students specialising in 192 – Construction and Civil Engineering. It is aimed at developing the knowledge and practical skills required for professional activities under conditions of extensive implementation of software packages in the construction industry.

The main objective of the discipline is to teach students how to apply modern information systems for the analysis and design of buildings, perform detailed modelling of reinforced concrete and steel structures, utilise specialised software packages in construction, address issues of computer modelling, and master the fundamentals of the finite element method. Particular attention is given to solving linear and nonlinear structural analysis problems, considering both engineering and physical nonlinearity, as well as the specific behaviour of structures of varying complexity.

During the course, students will:

- acquire knowledge of the principles for creating and analysing two-dimensional and three-dimensional structural models;
- learn to apply shell, frame, and solid models in numerical analysis;
- master techniques for accounting for the real behaviour of materials, including cracking, creep, and plastic deformations;
- develop skills in interpreting and critically evaluating calculation results, comparing them with regulatory requirements, and applying them to practical engineering solutions.

The practicum has been developed in accordance with the curriculum of the discipline. It is intended for use by students during practical classes and independent study. The materials include task execution algorithms, input data for modeling, examples of calculation schemes, and recommendations for result analysis.

An important component of mastering the discipline is the independent work of students, aimed at studying theoretical material, reinforcing practical skills in working with software packages, and developing the ability to apply acquired knowledge in real-world design conditions.

Thus, the study of the discipline “Modern Computer Methods for Structural Analysis” ensures the training of highly qualified specialists capable of using modern software and computational technologies for the analysis, design, and reliability assessment of building structures. This is a necessary condition for the development of the construction industry in the context of modern challenges.

All practical tasks presented are based on official examples from the LIRA-FEM software package, a domestic engineering tool that implements modern finite element method algorithms. This package is widely used in professional engineering practice both in Ukraine and abroad, ensuring compliance with current regulatory documents of Ukraine (DBN, DSTU), as well as the European Union standards (Eurocodes) in terms of structural analysis.

This approach enables students to develop practical skills in working with professional software that meets current international requirements and is useful in both local and global engineering practice.

PRACTICAL SESSION 1

MODELLING AND ANALYSIS OF A REINFORCED CONCRETE SLAB (LIRA-FEM)

The purpose of this task is to develop and reinforce practical skills in constructing a computational model of a slab, ensuring the correct application of boundary conditions, which is essential for the accuracy of subsequent engineering calculations. The work involves mastering the methodology for applying loads and forming the structural analysis model (SAM) to enable precise evaluation of the structure.

An additional objective is to acquire proficiency in reinforcement design using two methods – Karpenko's and Wood's approaches – followed by a comparison of the results to select the most efficient solution. The task also includes developing skills in analysing and interpreting graphical outputs, such as deformation maps, bending moment diagrams, and required reinforcement area (As).

Initial Data

1. Slab dimensions: 3 × 6 m, thickness 150 mm.
2. Support scheme: one long edge – simply supported along its entire length; opposite short edge – supported at the ends (columns); long side edges – free.
3. Mesh division: 6 × 12 (spacing 0.5 m in both directions).
4. Loads:

№ 1 – self-weight;

№ 2 – concentrated forces P = 1 t, P = 1 t, P = 1 t at nodes (see task diagram);

№ 3 – concentrated loads on plates P = 1 t, P = 1 t, P = 1 t with application area A = B = 0.25 m, A = B = 0.25 m, A = B = 0.25 m for the specified elements.

5. Two design approaches: Karpenko and Wood (slabs).
6. Materials: concrete C16/20, reinforcement A400C (longitudinal), A500C (transverse).

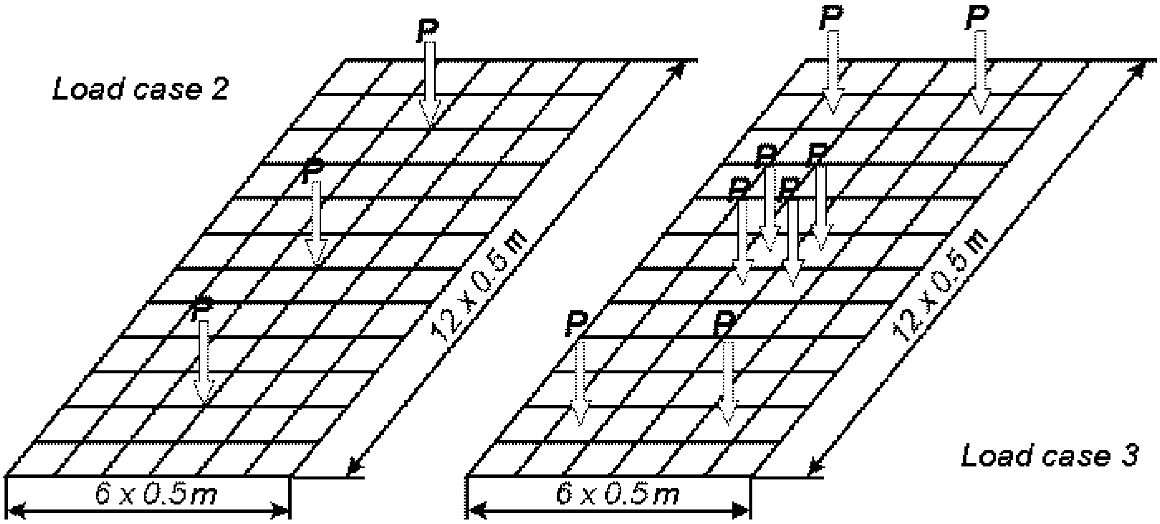


Figure 1 – Structural analysis scheme of the slab

Execution Algorithm

Task preparation

1. Create a new project: Programs → New; set the scheme flag: 3 DOF per node (Wz, Ux, Uy) in the X_0Y plane; give the file a name, e.g. 02_slab_RC.
2. Save the model (Data/...).

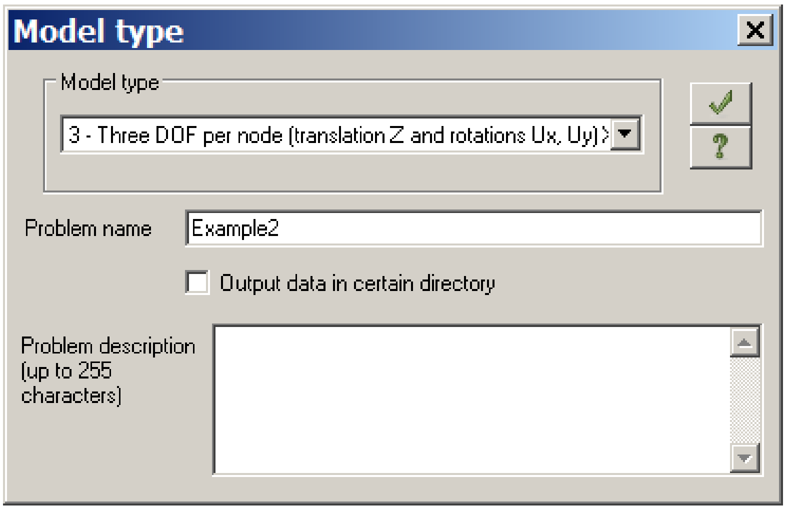


Figure 1.1 – Project description

Geometry and mesh

3. Generate the slab:

Create → Generate regular fragments/meshes → Slab; mesh spacing: 0.5 m in both directions; number of divisions: 6 × 12. → Apply.

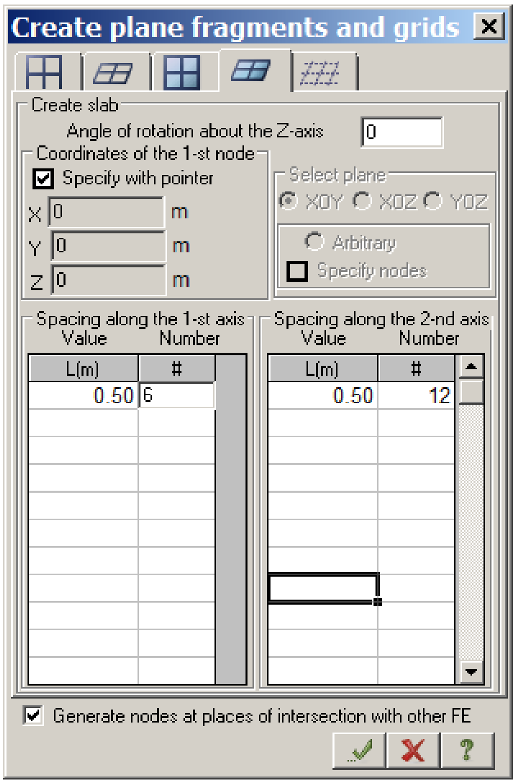


Figure 1.2 – Creation of planar fragments and meshes

Boundary conditions

4. Display node numbers; mark the support nodes (the entire long edge and the two corner nodes of the opposite short edge).

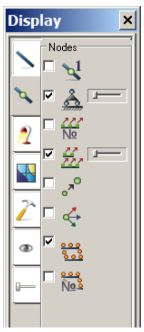


Figure 1.3 – “Show” dialog Figure

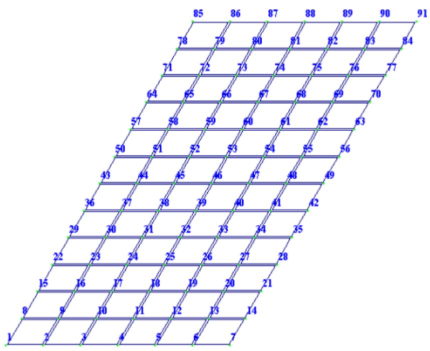


Figure 1.4 – Node numbering of the analysis scheme of the slab

5. Assign restraints (prohibit W_z for the marked nodes). Verify colour indication of the restraints.

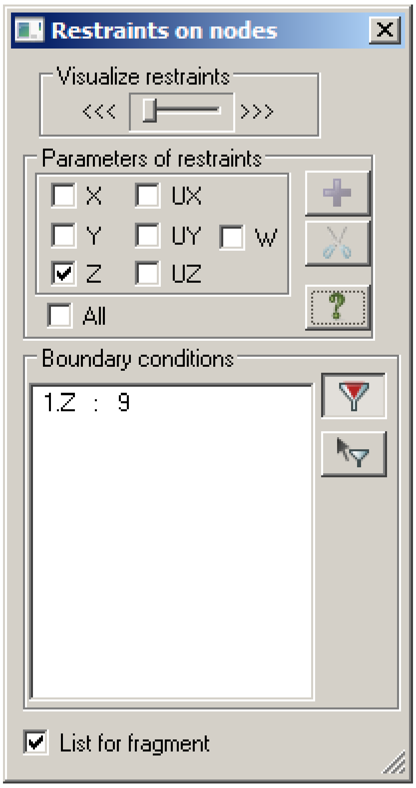


Figure 1.5 – “Node restraints” dialog

Design variants (reinforced concrete)

6. Open Design variants and create:

- Variant 1: DSF, type “Slab”, Karpenko theory.
- Variant 2: DSF, type “Slab”, Wood theory (tick option “Design reinforcement by Wood’s theory”).

7. In Stiffnesses and materials create a typePlate, h = 15 cm”; set $E \approx 3.0 \times 10^6$ t/m², **,** $\nu = 0.2$**,** $\rho \approx 2.75$ t/m³.

8. On the RC tab for the plates set: Calculation type*:* Slab, concrete class B20, reinforcement A–400C (longitudinal), A–240C (transverse).

9. Assign the selected stiffnesses and materials to all elements (first for the Wood variant, then switch to the Karpenko variant and assign its materials).

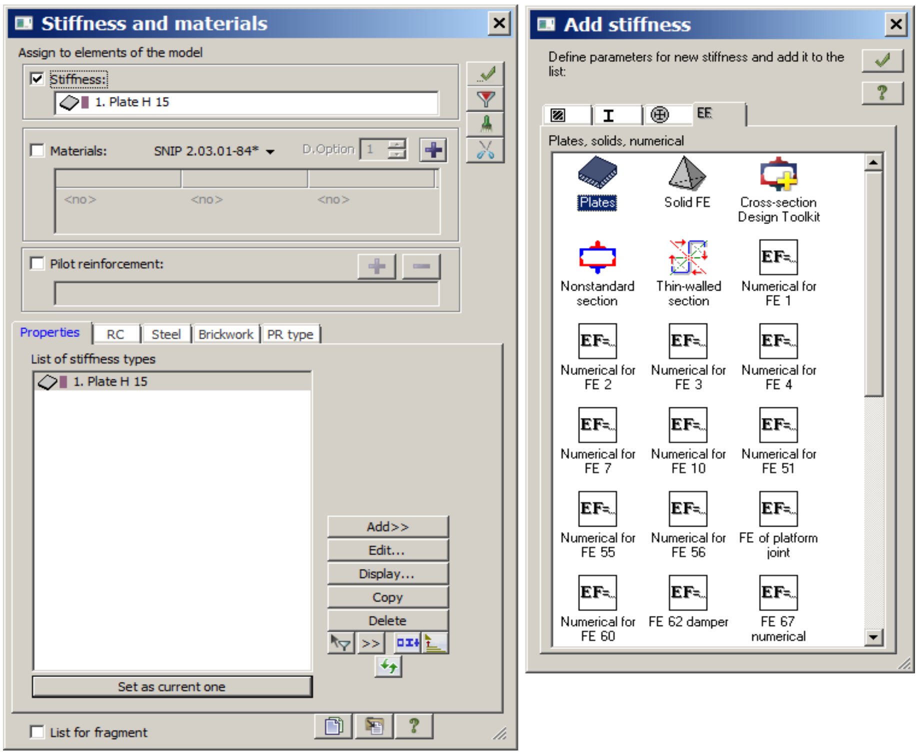


Figure 1.6 – “Stiffnesses and materials” (left) and “Add stiffness” (right) dialogs

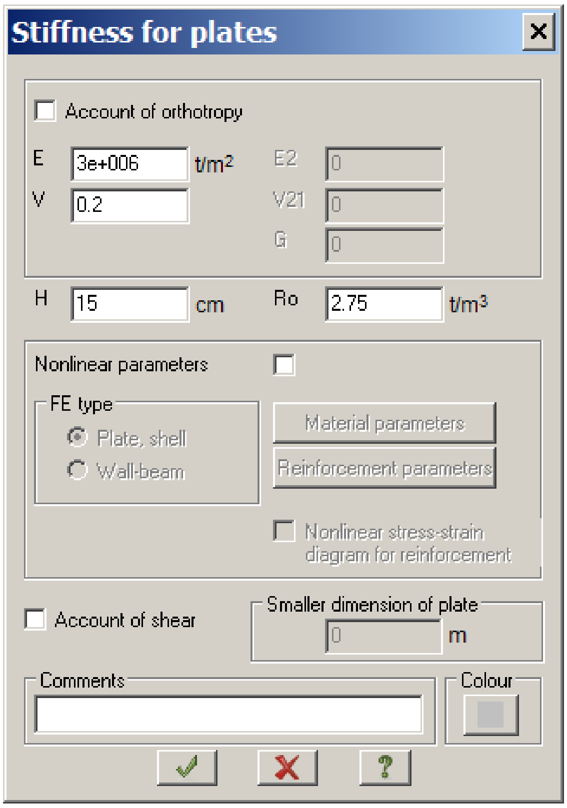


Figure 1.7 – “Plate stiffness definition” dialog

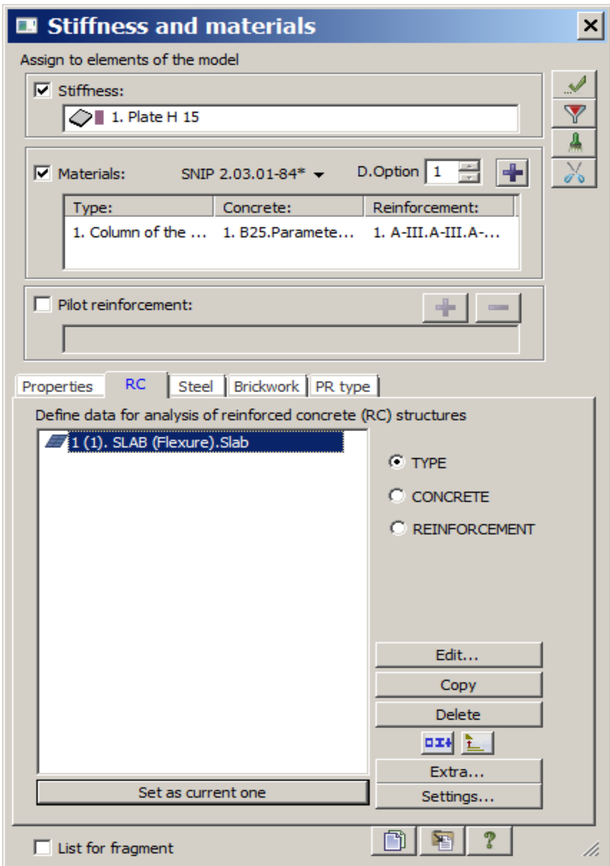


Figure 1.8 – “Stiffnesses and materials” dialog

Loads

10. Load № 1 (self-weight): Loads → Add self-weight ($\gamma_f = 1$, apply to “all”).

11. Load № 2 (nodal P = 1 t): select the specified nodes; Nodal loads → Concentrated force in Z (magnitude 1 t with negative sign in global Z if downward is negative).

12. Load № 3 (on plates P = 1 t): select the specified plate elements; Plate loads → Concentrated load with A = B = 0.25 m for the contact area.

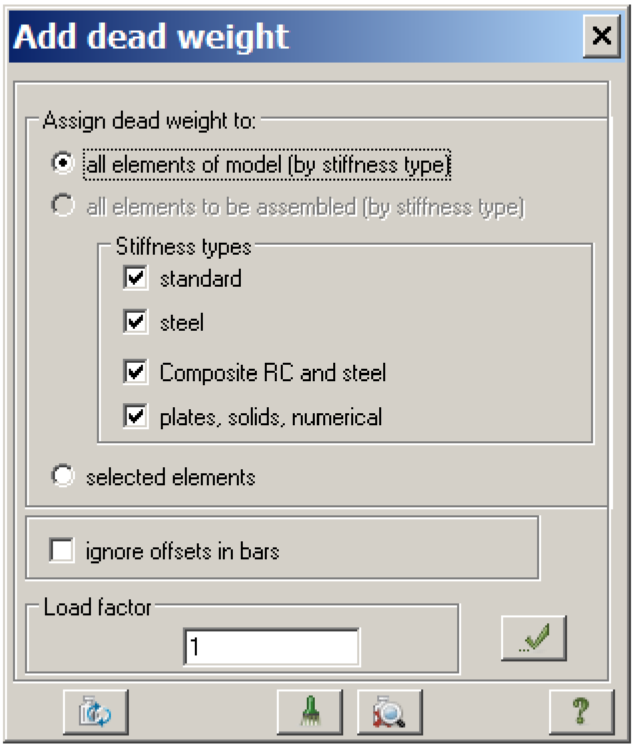


Figure 1.9 – “Add self-weight” dialog

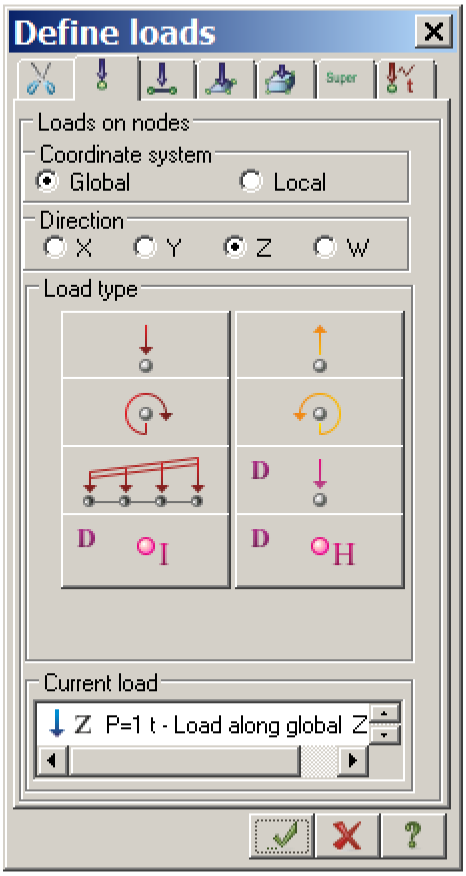


Figure 1.10 – “Load definition” dialog

13. In the Load editor as sign load types: № 1 – Permanent, № 2 and № 3 – Long-termvariable.

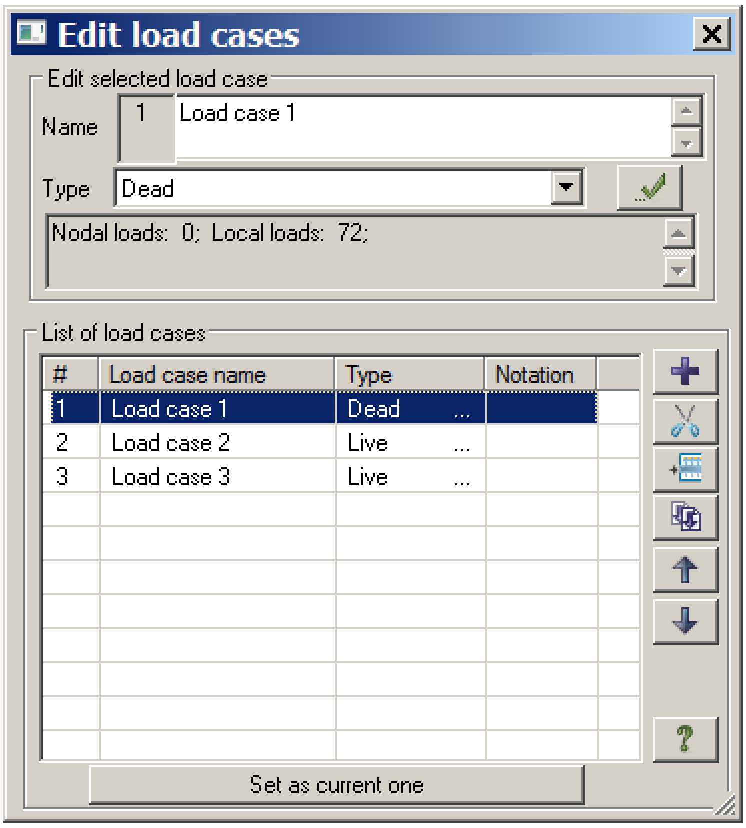


Figure 1.11 – “Load editor” dialog

Load combinations, calculation, analysis

14. DCF: Calculation → Table of load cases / structural analysis models (DCL/DCF) → “Fill by default” (check coefficients for each load type and adjust if necessary).

15. Full calculation: Calculation → Execute full analysis.

16. Review results (Analysis):

- Deformations (W_z isoplot).
- Mosaics of M_x,M_y for the plates.
- Standard tables (DCF in plates) → export to report.

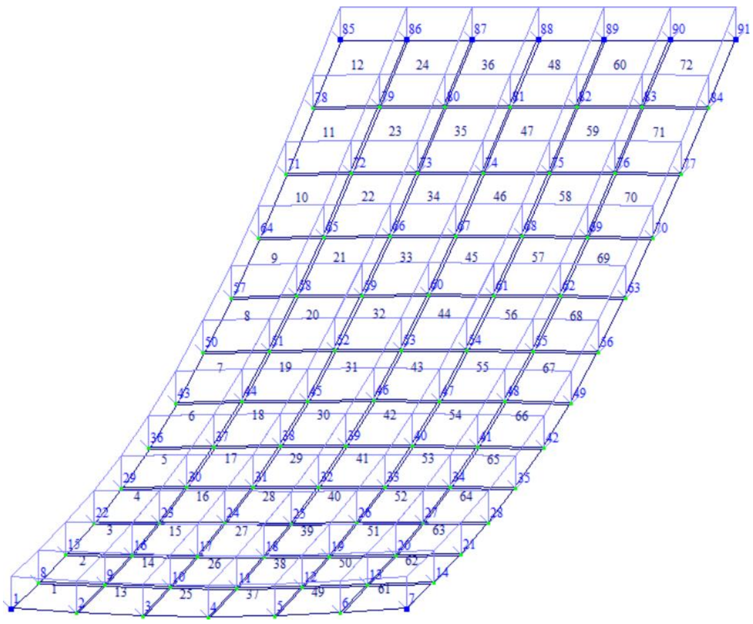


Figure 1.12 – Analysis scheme showing node displacements

Reinforcement design and comparison of theories

17. Reinforced concrete → Plate reinforcement: inspect the bottom reinforcement area As along X1 and Y1; for a particular element use Element information.

18. Generate reinforcement tables for the plates (for Variant 1 and Variant 2).

19. Switch between design variants and compare the As maps (Karpenko vs Wood) – note differences in peak zones (near supports / under concentrated loads).

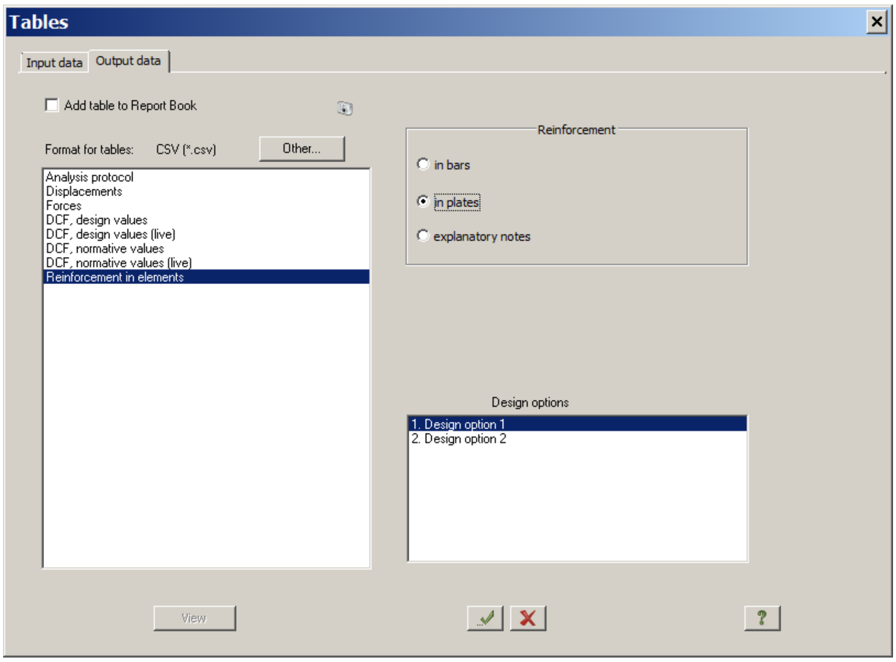


Figure 1.13 – “Tables” dialog

Tip: before comparing, hide node/element numbers and loads for clearer isoplot visualisation; unify the mosaic (contour) scales.

Typical errors

- Confusion over the Z-axis sign (load direction).
- Inconsistent assignment of materials across design variants.
- Incorrect identification of nodes/elements when assigning P = 1 t.
- Mismatched load types → erroneous load combination table.
- Comparing contour maps with different scales.

Self-check questions

1. How does the support scheme (free edge vs line/point support) affect the distribution of M_x and M_y?
2. Why are peak As values expected near supports / under concentrated loads?
3. How do the reinforcement design results differ between Karpenko's and Wood's theories for this slab, and why?
4. Which load combination produced the maximum moments? Where are their maxima and minima located?
5. Why is it important to standardise mosaic scales when comparing design variants?
6. How will the W_z, M_x, $M_y$andAs patterns change if the nodal concentrated forces are replaced by uniformly distributed loads over the contact area?

PRACTICAL SESSION 2

ANALYSIS OF A TWO-DIMENSIONAL REINFORCED CONCRETE FRAME (LIRA-FEM)

The objective of this task is to acquire and reinforce practical skills in creating a computational model of a planar reinforced concrete frame, taking into account supports, nodes, and structural elements. The exercise involves mastering the methodology for defining different types of loads – nodal and distributed on beams – as well as forming the calculation model for subsequent analysis.

An additional aim is to perform a full calculation and select reinforcement for the frame elements in compliance with current regulatory requirements. Special attention is given to developing skills in interpreting analysis results, particularly in evaluating displacements, determining internal forces, and assessing reinforcement requirements.

Input Data

1. Scheme: two-span, two-storey reinforced concrete frame in the X0Z plane.
2. Spans: $L_1 = L_2 = 6.0$ m.
3. Storey heights: $H_1 = H_2 = 3.0$ m.
4. Cross-sections:
   - Columns: 400 × 400 mm;
   - Beams: 300 × 500 mm.
5. Materials: concrete C20/25, reinforcement A400C.
6. Node grid: according to the span lengths and storey heights.
7. Boundary conditions: rigid fixity at column bases.
8. Loads:
   - Self-weight of the structure.
   - Temporary uniformly distributed load on beams.
   - Temporary concentrated loads at selected nodes.

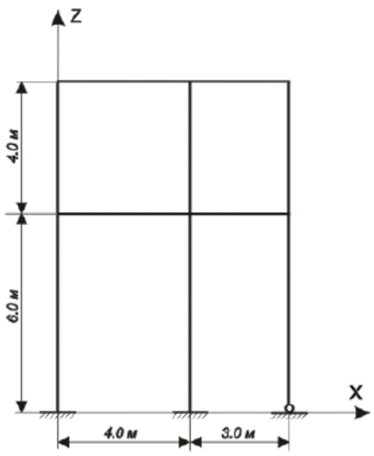


Figure 2.1 – Frame scheme

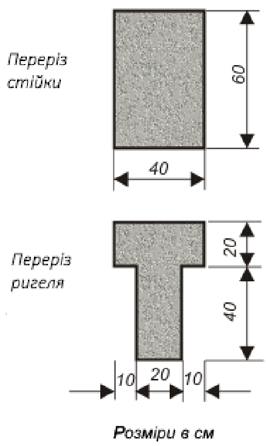


Figure 2.2 – Cross-sections of frame elements

Execution Algorithm

Task Preparation

1. Create a new project: Programs → New, set DOF = 3 (U_x, U_γ, φ) in the X0Z plane.

2. Save the file under the name Frame_2D.

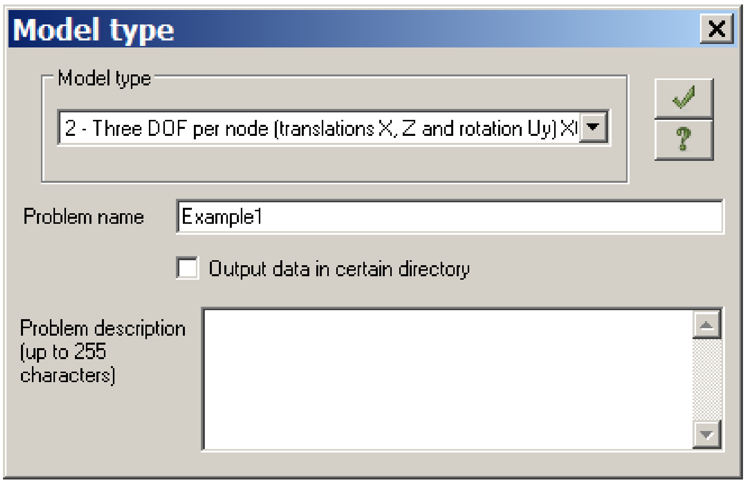
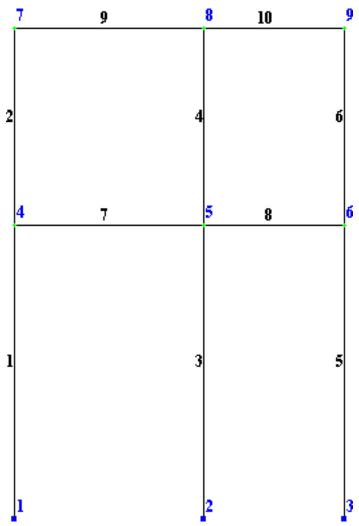


Figure 2.3 – Scheme description

Geometry Modelling

3. Create nodes at coordinates according to span lengths and storey heights.
4. Connect the nodes with bar elements: beams and columns.
5. Assign geometric properties to the elements:
   - Stiffness → Bar → h, b according to the given cross-sections.



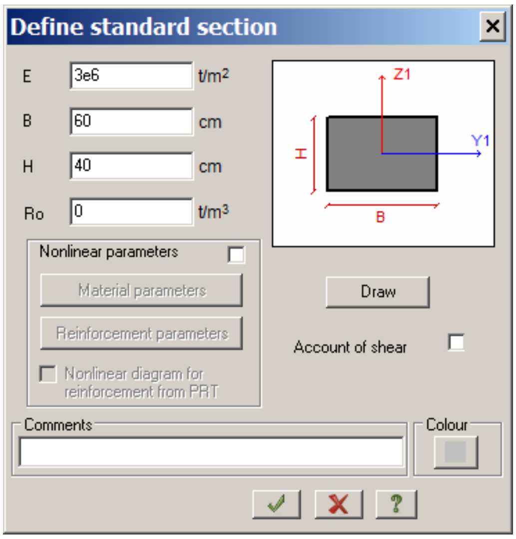


Figure 2.4 – Analysis model

Figure 2.5 – Assigning standard cross-section

Materials and Design

6. Create a material type: concrete C20/25, reinforcement A400C.

7. In Design variants create DCF for elements of type “Beam” and “Column” with the respective concrete and reinforcement classes.

8. Assign materials to the elements.

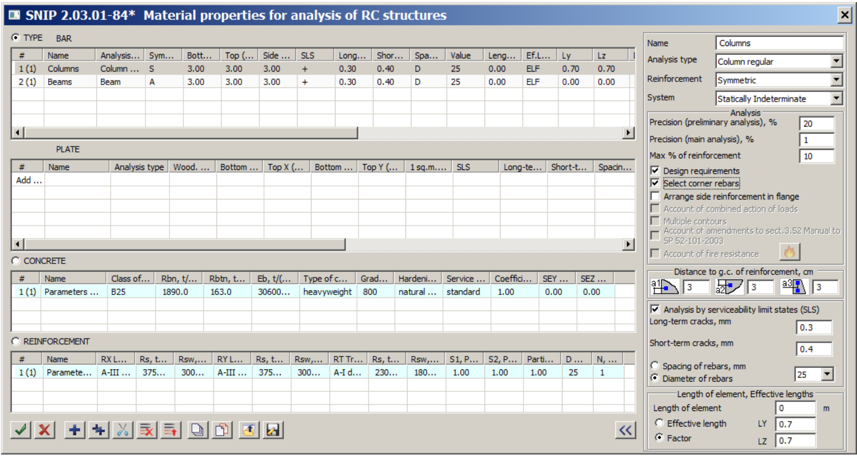


Figure 2.6 – Materials and design settings

Boundary Conditions

9. Assign rigid fixity to the nodes at the bases of the columns (restrain U_x, U_y, φ).

Loads

10. Self-weight – automatically added (γf = 1, Permanent).

11. Uniformly distributed load – applied to beam elements (Long-term variable).

12. Concentrated forces – applied at selected nodes (Long-term variable).

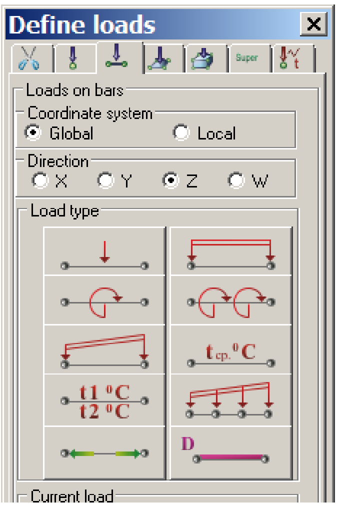


Figure 2.7 – Load definition

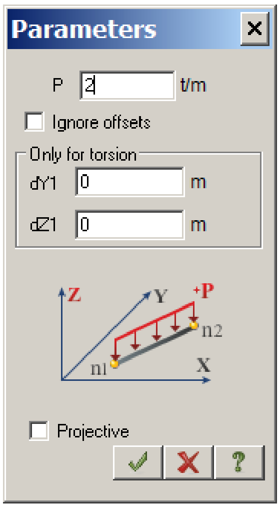
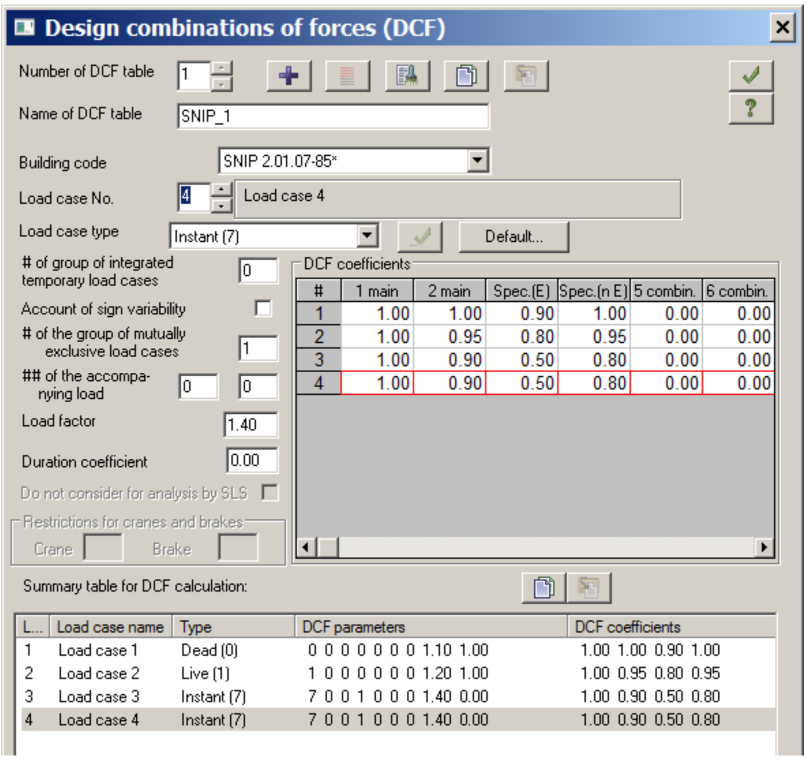


Figure 2.8 – Load parameters input

Load Combinations and Analysis

13. Fill the LC table using the default option, checking coefficients.



#	1 main	2 main	Spec.(E)	Spec.(n E)	5 combin.	6 combin.
1	1.00	1.00	0.90	1.00	0.00	0.00
2	1.00	0.95	0.80	0.95	0.00	0.00
3	1.00	0.90	0.50	0.80	0.00	0.00
4	1.00	0.90	0.50	0.80	0.00	0.00

L...	Load case name	Type	DCF parameters	DCF coefficients
1	Load case 1	Dead (0)	0 0 0 0 0 0 0 1.10 1.00	1.00 1.00 0.90 1.00
2	Load case 2	Live (1)	1 0 0 0 0 0 0 1.20 1.00	1.00 0.95 0.80 0.95
3	Load case 3	Instant (7)	7 0 0 1 0 0 0 1.40 0.00	1.00 0.90 0.50 0.80
4	Load case 4	Instant (7)	7 0 0 1 0 0 0 1.40 0.00	1.00 0.90 0.50 0.80

Figure 2.9 – Calculation combinations table

14. Perform the full calculation.

15. Review results: displacements, bending moments (M), shear forces (Q), and axial forces (N) in elements.

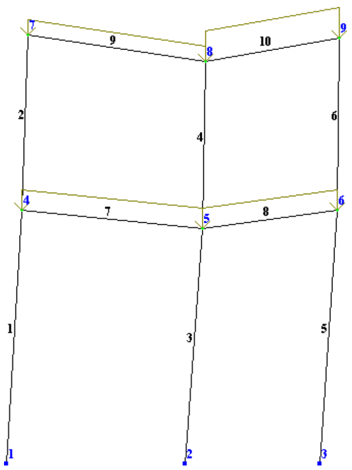


Figure 2.10 – Analysis model with node displacements

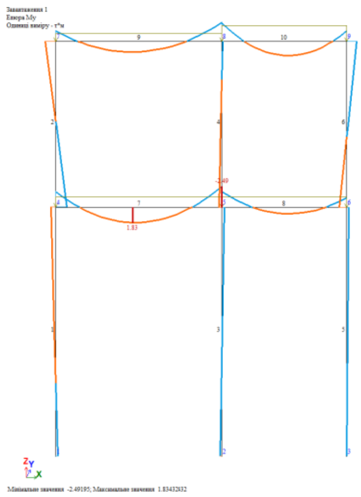


Figure 2.11 – Bending moment diagrams (M_u)

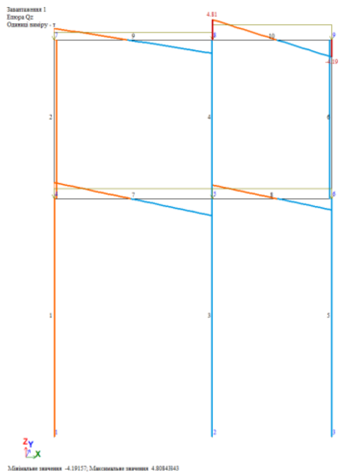


Figure 2.12 – Shear force diagrams (Q_z)

Reinforcement Design

16. For columns and beams, review As maps (longitudinal and transverse reinforcement).

17. Compare internal forces between spans and storeys.

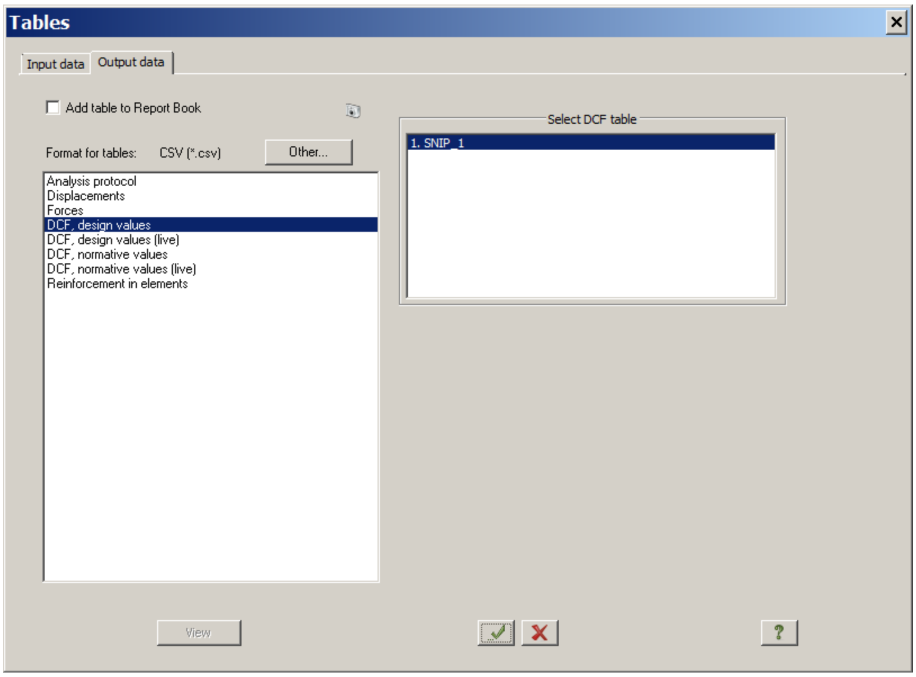


Figure 2.13 – Generating and viewing result tables

Results Analysis

18. Identify maximum bending moments and shear forces.

19. Draw conclusions about the frame behaviour.

Self-Check Questions

1. How does changing the storey height affect M and Q values in beams?

2. What are the specific features of a rigidly fixed frame compared to a simply supported one?

3. In which elements of the frame do the largest axial forces (N) occur, and why?

4. How is reinforcement distributed in the top and bottom zones of the beam under uniform loading?

5. Why is it important to separate permanent and temporary loads when forming load combinations?

PRACTICAL SESSION 3

ANALYSIS OF A STEEL TOWER (LIRA-FEM)

The aim of this task is to master the methodology for creating a spatial bar model of a steel tower, taking into account its structural components, such as bracing members, chords, and bracing systems that ensure the overall spatial stiffness of the structure. The exercise includes learning the correct procedure for applying spatial loads, including the self-weight of the structure and the effects of wind loading.

Additionally, the task involves calculating the stress-strain state (SSS) of the steel tower and subsequently selecting the optimal cross-sections for its members in accordance with design standards. An important component of the exercise is developing the ability to interpret analysis results, including displacement evaluation, determination of internal forces, and stability verification.

Initial Data

- Structure: four-sided spatial bar tower.
- Height: H = 30 m.
- Number of sections: 6 (each 5 m high).
- Base geometry**:** square, 4 × 4 m.
- Elements:
  1. Vertical legs – circular hollow sections Ø159 × 6 mm (S355).
  2. Horizontal and diagonal bracings – circular hollow sections Ø89 × 4 mm (S355).
- Boundary conditions: hinged support at the base.
- Loads:
  1. Self-weight (automatically applied).
  2. Wind load according to national standards/Eurocode (e.g., q_o = 0.3 kPa at 10 m height, including relevant coefficients).
  3. Additional concentrated load on the top platform (antennas, equipment).

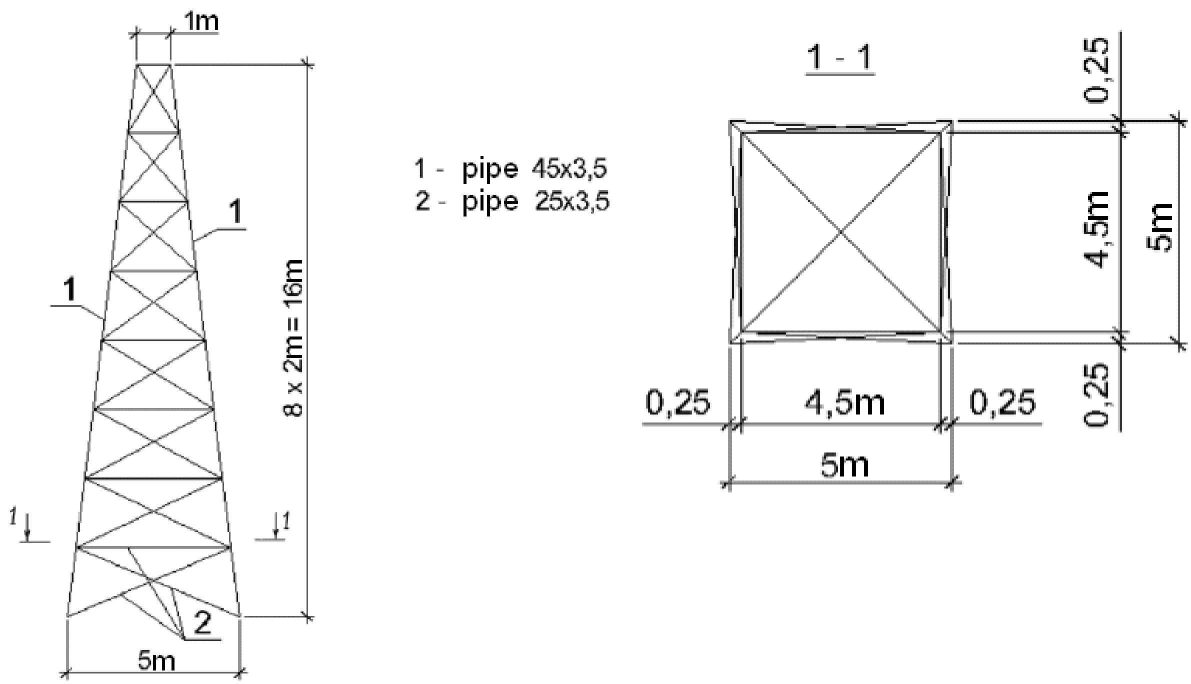


Figure 3.1 – Tower layout

Execution Algorithm

Task Setup

1. Create a new project: Spatial frame, DOF = 6.
2. Assign the name Steel_Tower.

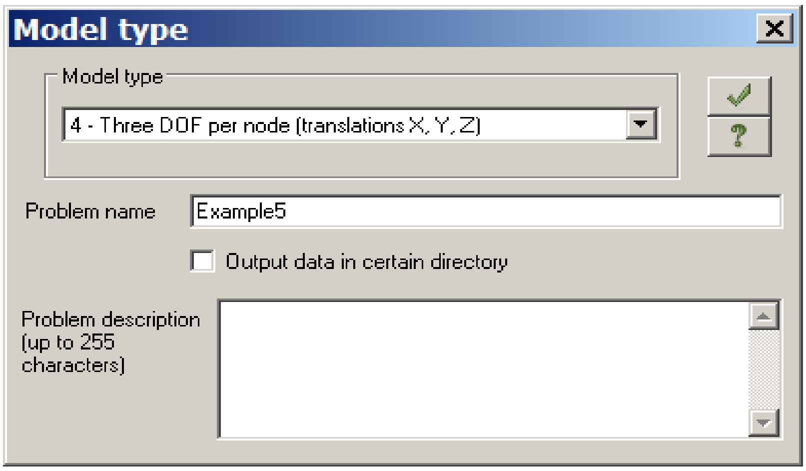


Figure 3.2 – Scheme description

Geometry Modelling

3. Define the base nodes (4 points forming a 4 × 4 m square).
4. Copy the nodes upwards in 5 m steps to a height of 30 m.

5. Connect the vertical legs between corresponding nodes at the corners.

6. Add horizontal bracing members (around the perimeter of each section).

7. Add diagonal bracing members in the planes of the tower faces.

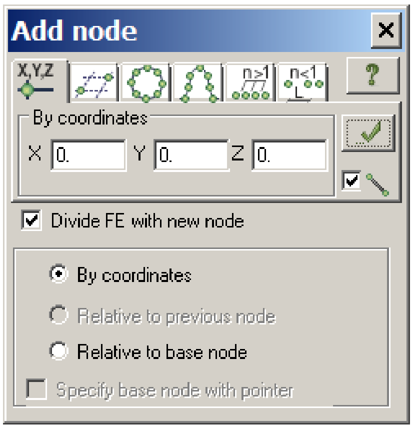


Figure 3.3 – Base nodes

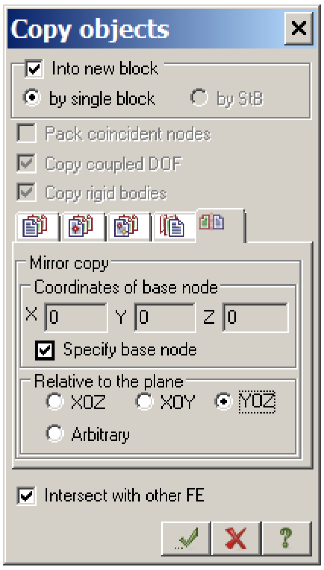


Figure 3.4 – Copying objects

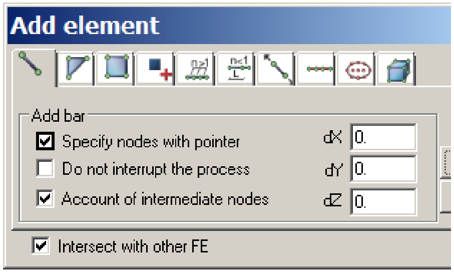


Figure 3.5 – Adding members between nodes

Materials and Cross-Sections

8. Create material S355 (define modulus of elasticity, density).

9. Assign section properties for the legs, horizontal, and diagonal bracing members.

10. Check the orientation of local axes.

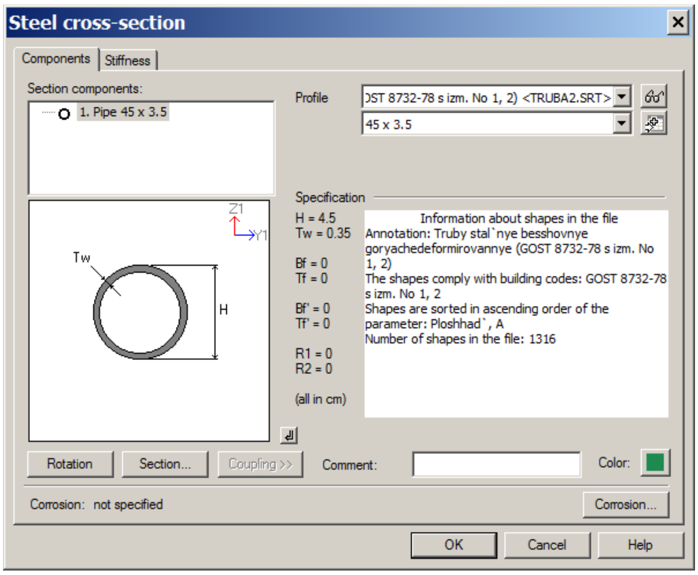


Figure 3.6 – Defining cross-sections

Supports

11. Assign hinged supports at the base nodes (restrain U_x, U_γ, U_z, φ_x, φ_γ, φz).

Loads

12. Permanent load: self-weight (γf = 1, Permanent).

13. Wind load: apply a linearly distributed load along the height to the legs, considering the variation of pressure with height.

14. Concentrated load: applied to the top nodes (equipment).

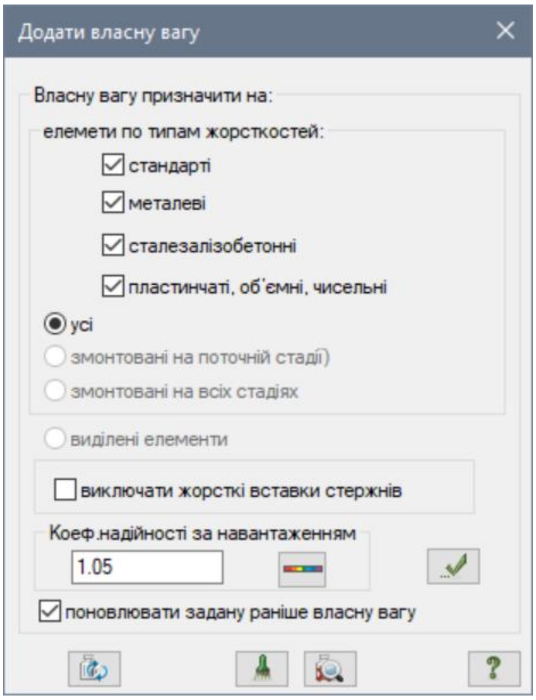


Figure 3.7 – Self-weight definition

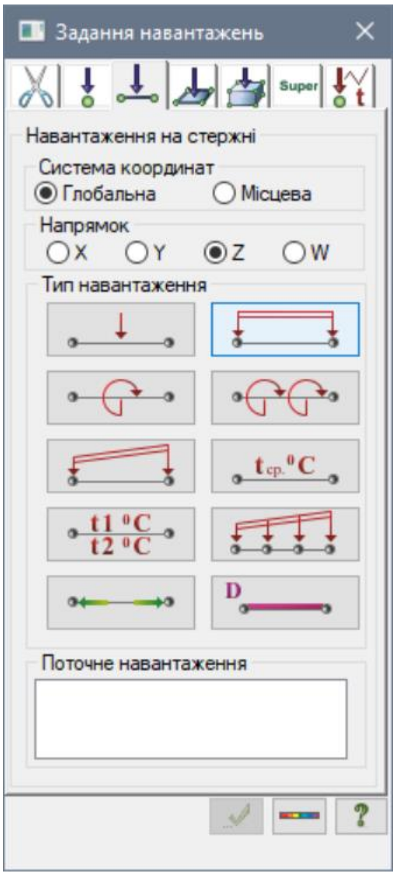


Figure 3.8 – Load application

Load Combinations and Analysis

15. Create load combinations according to Eurocode.

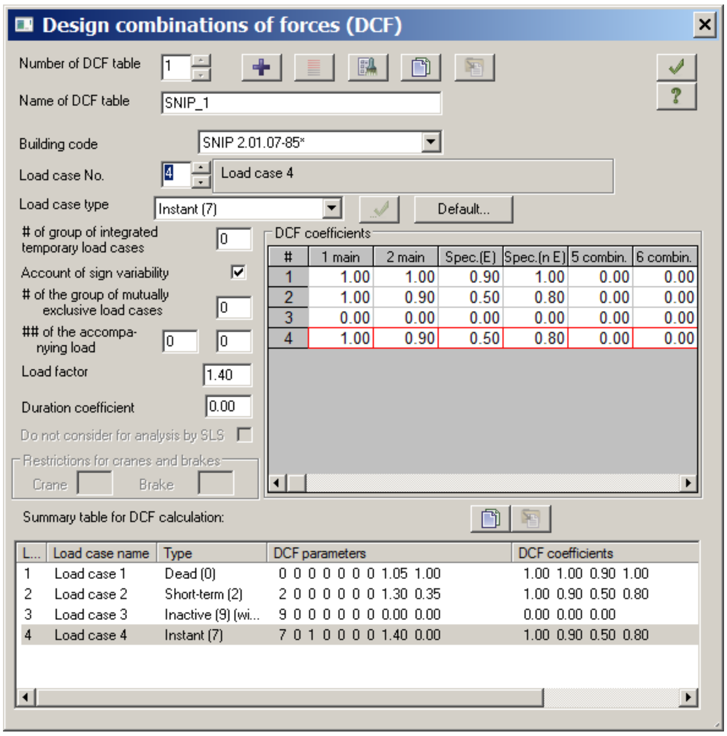


Figure 3.9 – Load combination table

16. Perform the stress-strain state analysis.

17. Review results: displacements, bending moments (M), axial forces (N), and shear forces (Q).

Section Verification and Design

18. Perform section checks according to standards for tension, compression, and bending members.

19. Verify utilisation factors (≤ 1).

20. Determine the maximum horizontal displacement at the tower top.

21. Identify the most heavily loaded members.

22. Draw conclusions regarding structural stability and efficiency.

Self-Check Questions

1. How does the horizontal displacement of the tower change with increasing height?
2. Which members primarily resist wind loads?
3. How do diagonal bracings influence structural stiffness?
4. What is a section utilisation factor and how should it be interpreted?
5. Why is it important to consider wind pressure variation with height?

PRACTICAL CLASS 4

ANALYSIS OF A PLANAR STEEL FRAME (LIRA-FEM)

The aim of the task is to master the methodology of constructing a planar bar model of a steel frame, taking into account the spans, heights, and stiffness characteristics of its elements. Within the scope of the work, students are expected to acquire the principles of correctly defining various types of loads for a planar model, including permanent, variable, snow, and wind loads.

Another important stage is the calculation of the stress–strain state of the structure and the verification of the strength of its elements in accordance with regulatory requirements. The task provides for the development of skills in interpreting results, including the assessment of displacements, determination of internal forces, and selection of optimal cross-sections of the steel frame elements.

Initial Data

1. Structure: single-span planar frame.
2. Span: L = 24 m.
3. Column height: 6 m.
4. Material: steel S355.
5. Elements:
   - Columns – I-beam IPE 300.
   - Beam – I-beam IPE 400.
6. Supports: pinned at the column bases.
7. Loads:
   - Self-weight of the structure.
   - Permanent load from the roof covering: 0.8 kN/m².
   - Snow load according to DBN/Eurocode.
   - Wind load (for planar model – horizontal concentrated or distributed).

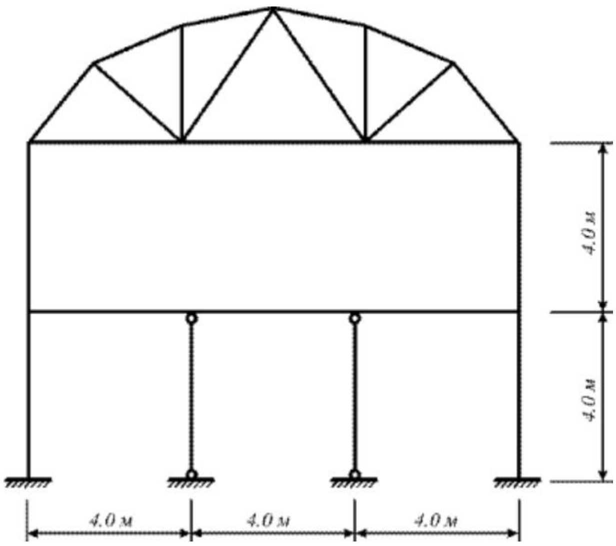


Figure 4.1 – Cross-section scheme of the building

Execution Algorithm

Task creation

1. Create a new project “Planar frame”, DOF = 3.

2. Set the units of measurement – kN, m, °C.

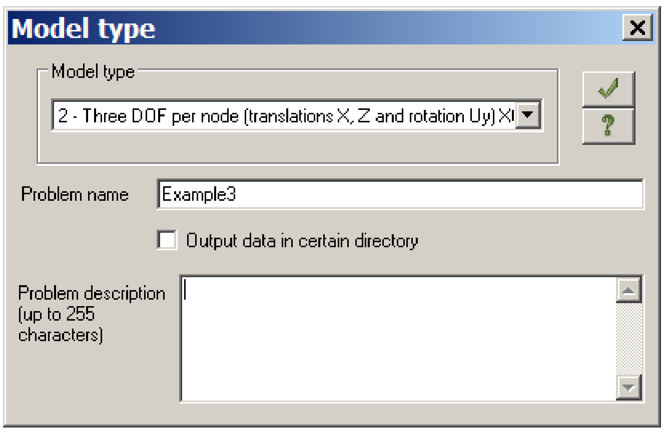


Figure 4.2 – Scheme description

Geometry

3. Define column nodes at coordinates (0;0), (L;0) and the top nodes of the columns at a height of 6 m.

4. Connect the columns and the beam into a single frame scheme.

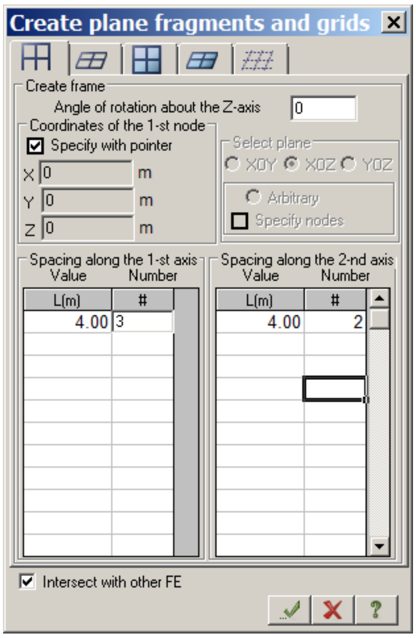


Figure 4.3 – Creation of the geometric scheme

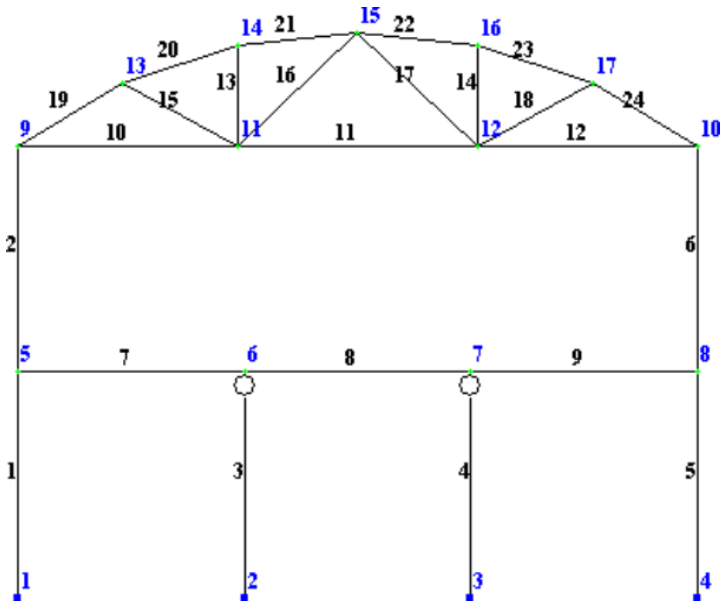


Figure 4.4 – Calculation scheme

Materials and cross-sections

5. Create material S355 ($E = 2.1 \cdot 10^5$ MPa, $\gamma = 78.5$ kN/m³).

6. Assign cross-sections: IPE 300 for columns, IPE 400 for the beam.

Supports

7. At the base nodes of the columns – pinned support ($U_x = U_Y = \varphi z = 0$).

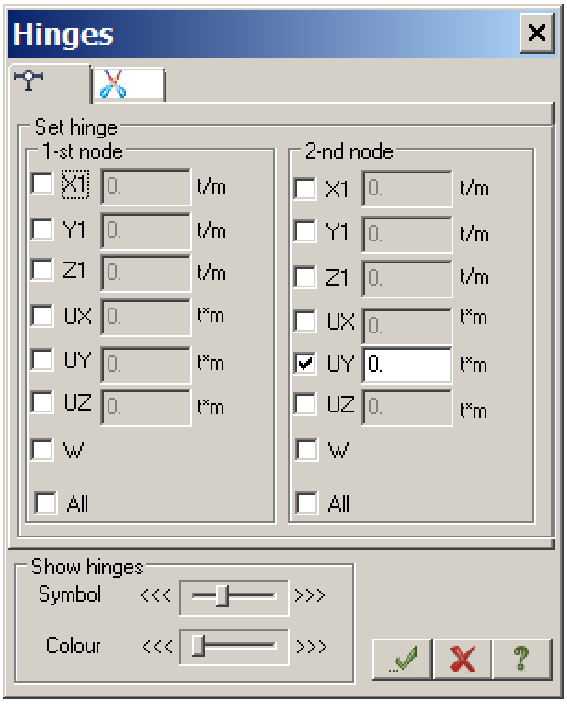


Figure 4.5 – Support assignment

Loads

8. Self-weight – automatic.

9. Permanent load – uniformly distributed along the beam.

10. Snow – calculate according to standards and apply along the beam.

11. Wind – horizontal load on the columns.

Calculation and Load Combinations

12. Create load combinations in accordance with Eurocode.

13. Perform stress–strain analysis.

14. Review displacements and internal forces M, N, Q.

Verification and cross-section selection

15. Perform verification according to standards (stability, strength).

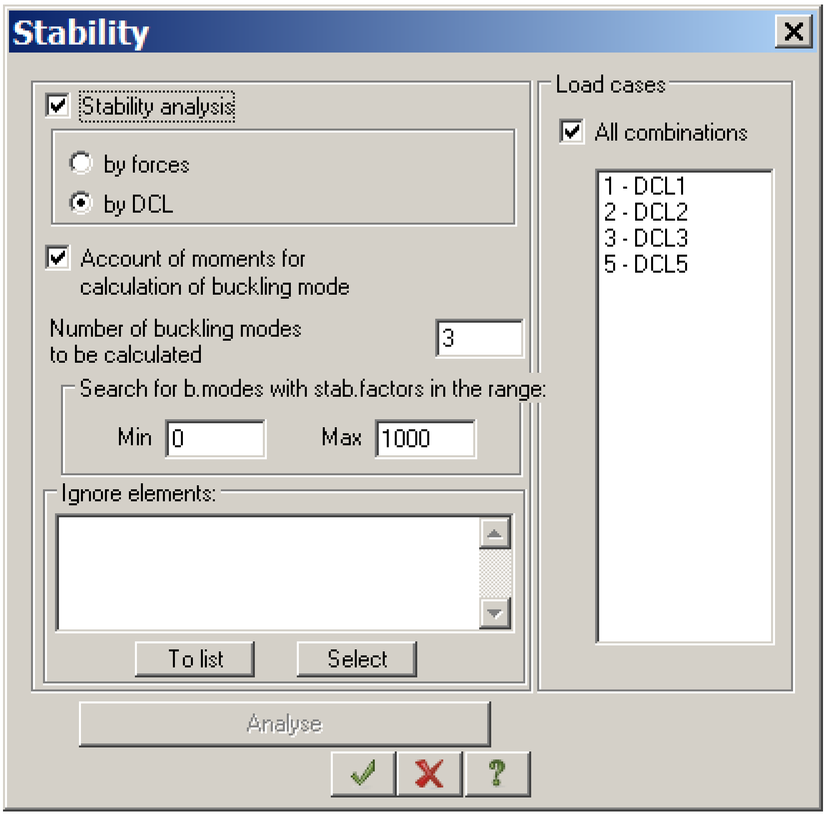


Figure 4.6 – Setting parameters for frame stability analysis

16. If necessary, change cross-sections to ensure a utilisation factor ≤ 1.

Result analysis

17. Determine the maximum deflection of the beam.

18. Analyse the distribution of bending moments in the column and the beam.

19. Draw conclusions regarding the adequacy of stiffness.

Self-assessment questions

1. How does the column height affect the beam deflection?
2. Why is the bending moment at the column base equal to zero with pinned supports?
3. How will the distribution of moments change with an increased span?
4. What does a cross-section utilisation factor of 0.95 mean?
5. Why is it important to verify both strength and stability of the elements?

PRACTICAL CLASS 5

ANALYSIS OF A CYLINDRICAL REINFORCED CONCRETE TANK

The aim of the task is to acquire practical skills in constructing a three-dimensional model of a thin-walled cylindrical shell in the LIRA-FEM software package, taking into account the specific features of the spatial behaviour of the structure. The work involves mastering the configuration of stiffness characteristics of shell elements for the correct modelling of the reinforced concrete walls of the tank.

Important stages include defining boundary conditions, applying loads from hydrostatic pressure and self-weight, as well as performing stress-strain analysis of the structure. The task also requires the interpretation of the obtained results, including the assessment of stresses, displacements, and internal forces in the tank wall, together with familiarisation with the principles of strength verification of shell structures in accordance with current standards.

Initial Data

• Internal diameter: 12 m.

• Wall height: 8 m.

• Wall thickness: 200 mm.

• Material: concrete grade C30/37 (E = 30,000 MPa, γ = 25 kN/m^3).

• Supports: rigid restraint at the nodes of the bottom ring of the wall.

• Loads:

1. Self-weight of the structure.

2. Hydrostatic water pressure varying along the height (from 0 at the top to maximum at the bottom).

Execution Algorithm

Task creation

1. Create a new project in the Shell Elements mode with measurement units – m, kN.

2. Specify the model parameters (analysis in a physically linear setting).

Geometry construction

3. Create a closed circle with a radius of 6 m in plan view.

4. Divide the circle into 36–72 segments (depending on the required accuracy).

5. Extrude (copy vertically) to a height of 8 m, dividing the height into 8–16 rings.

Note: the finer the mesh, the more accurate the analysis, but the greater the computational effort.

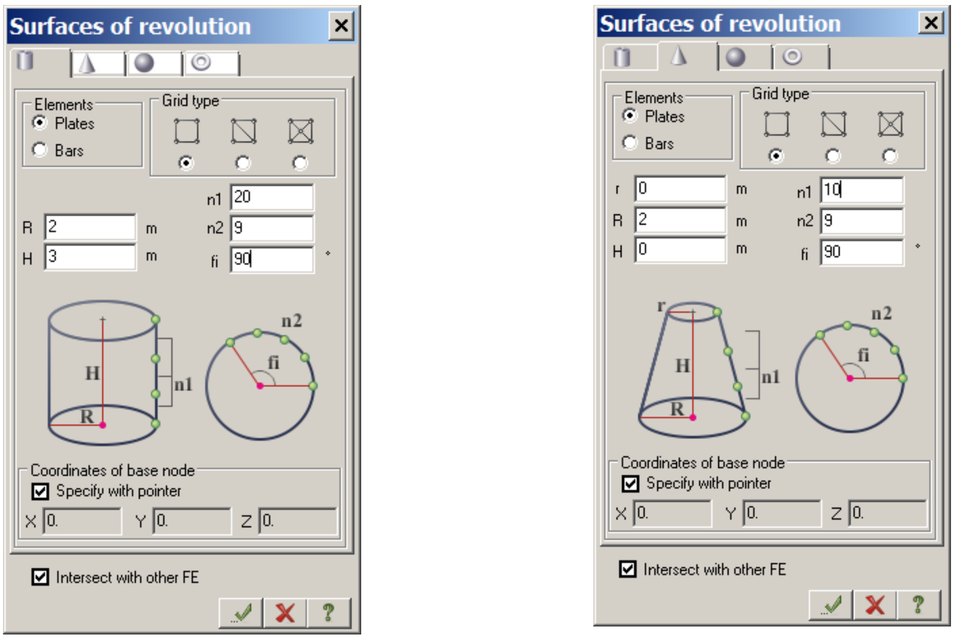


Figure 5.1 – Creation of the geometric scheme

Materials and stiffness assignment

6. Create material C32/40, assigning elasticity modulus, density, and Poisson's ratio.

7. Add a stiffness type "Shell" with a thickness of 0.2 m.

8. Assign this stiffness to all wall elements.

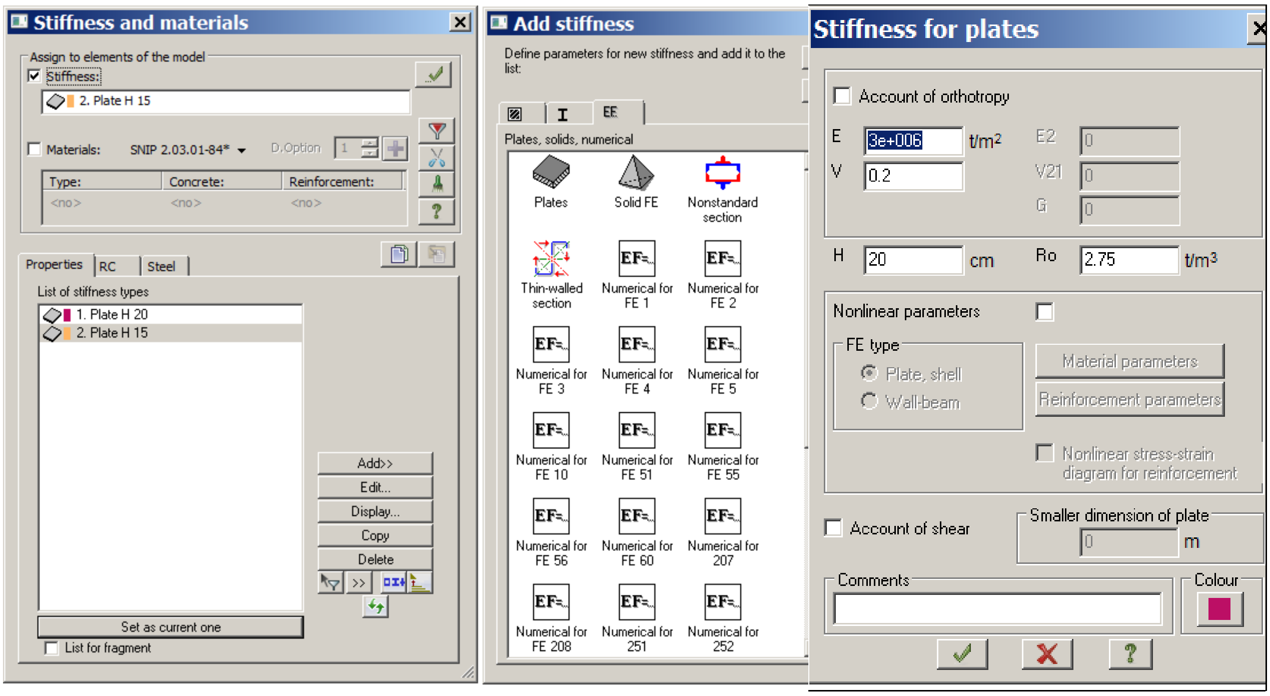


Figure 5.2 – Assignment of stiffness and material parameters to the tank wall elements

Boundary conditions

9. Fix displacements in X, Y, and Z directions at the nodes of the bottom ring.

10. Ensure compatibility of nodes in the direction of tangential displacements (if necessary).

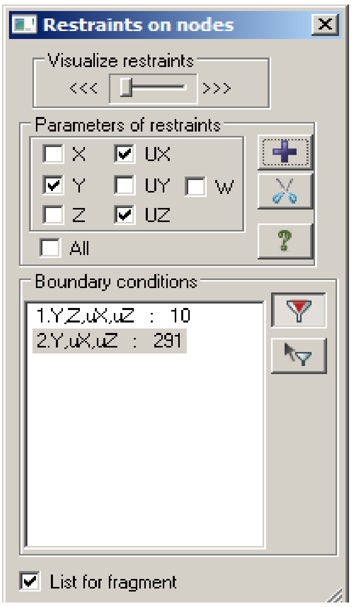


Figure 5.3 – Boundary condition assignment

Loads

11. Activate self-weight of the structure.

12. Define hydrostatic pressure load:

• Enable shell element mode.

• Create a load with linear variation of intensity along the height from 0 kN/m² (top) to γ × h (bottom).

13. Apply the load to the inner surface of the wall.

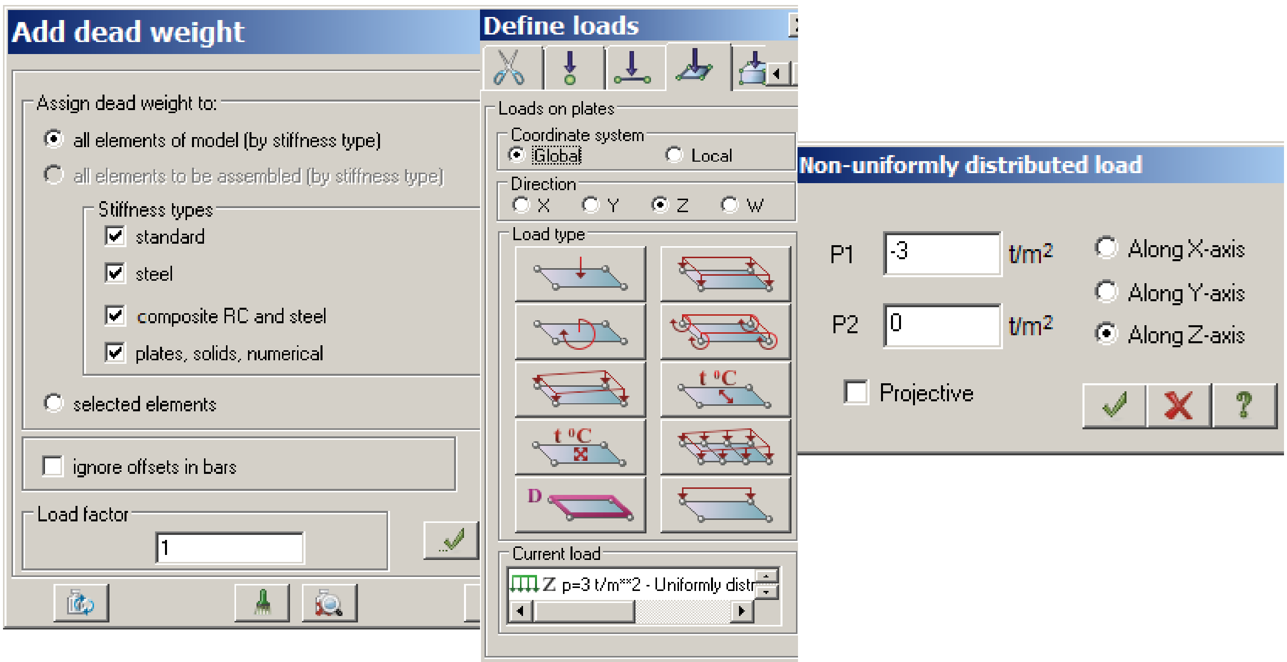


Figure 5.4 – Load assignment

Calculation and result analysis

14. Define load cases and combinations.

15. Perform the calculation.

16. Review:

- displacement maps (radial, vertical);
- membrane and bending stress maps;
- force distribution along the height.

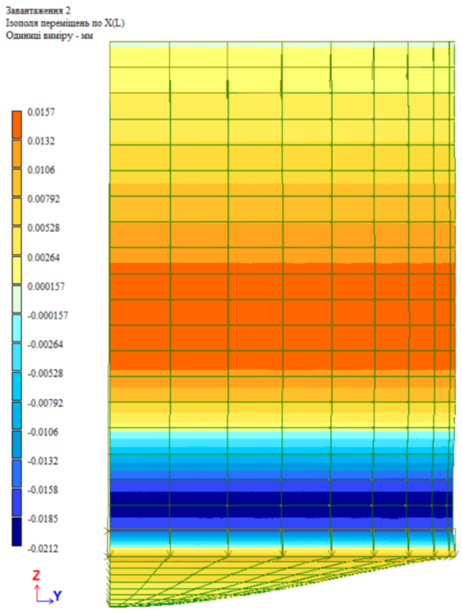


Figure 5.5 – Isolines of displacements along the local X-axis

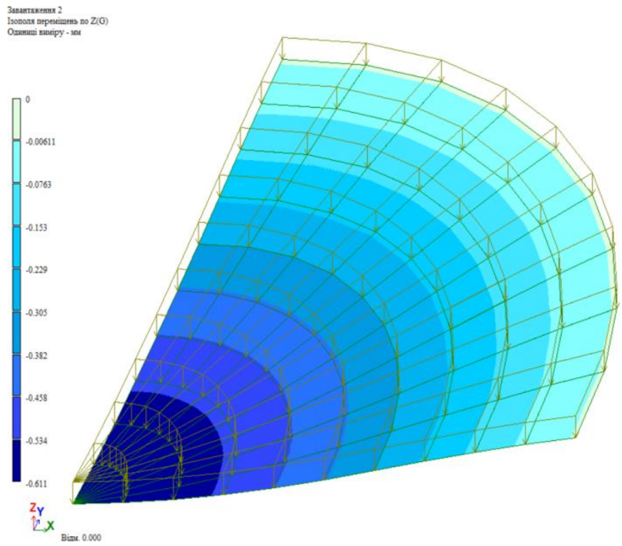


Figure 5.6 – Isolines of displacements along the global Z-axis

Strength verification

17. Perform verification of shell elements in accordance with standards (concrete in tension and compression).

18. Identify critical areas for reinforcement (region near the base, joints).

Self-assessment questions

1. How does the mesh density affect the accuracy of the shell analysis?

2. Why do the highest membrane stresses occur in the base region of the tank?

3. How will the stress distribution change if the wall thickness is increased?

4. What is the difference between membrane and bending stresses in shells?
5. Why is only the internal hydrostatic pressure considered in the analysis?

PRACTICAL CLASS 6

ANALYSIS OF MASTS IN A GEOMETRICALLY NONLINEAR SETTING

The aim of the task is to acquire practical skills in constructing a three-dimensional bar model of a mast in the LIRA-FEM software package, taking into account the specific behaviour of flexible structures. The work involves mastering the methodology of configuring geometrically nonlinear analysis, which allows the influence of deformations on the stress–strain state of the system to be considered.

Important stages include the application of wind loads, self-weight, and guy cables (stays), as well as analysing the effect of geometric nonlinearity on displacements and internal forces in the mast elements.

Initial Data

• Mast height: 40 m.

• Scheme: three tiers of stays, spacing along height – 13.3 m.

• Material: steel C245, $E = 2 \times 10^5$ MPa, $\gamma = 78.5$ kN/m³.

• Supports: rigid restraint of the nodes of the bottom ring of the mast.

• Loads:

1. Self-weight.
2. Wind load according to DBN or Eurocode.
3. Initial tensioning of stays.

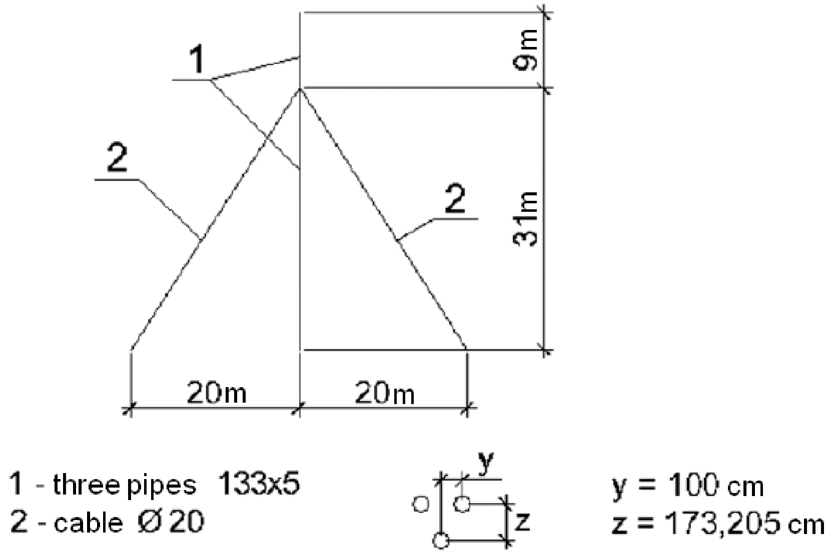


Figure 6.1 – Mast scheme

Execution Algorithm

Task creation

1. Select units: m, kN.

2. Enable geometrically nonlinear analysis in the settings.

Geometry construction

3. Create a vertical axis (line) with a height equal to that of the mast.

4. Divide it into segments according to the locations of the stay attachment nodes.

5. Create three tiers of stays, connecting the respective nodes with anchorage points in the ground at an angle.

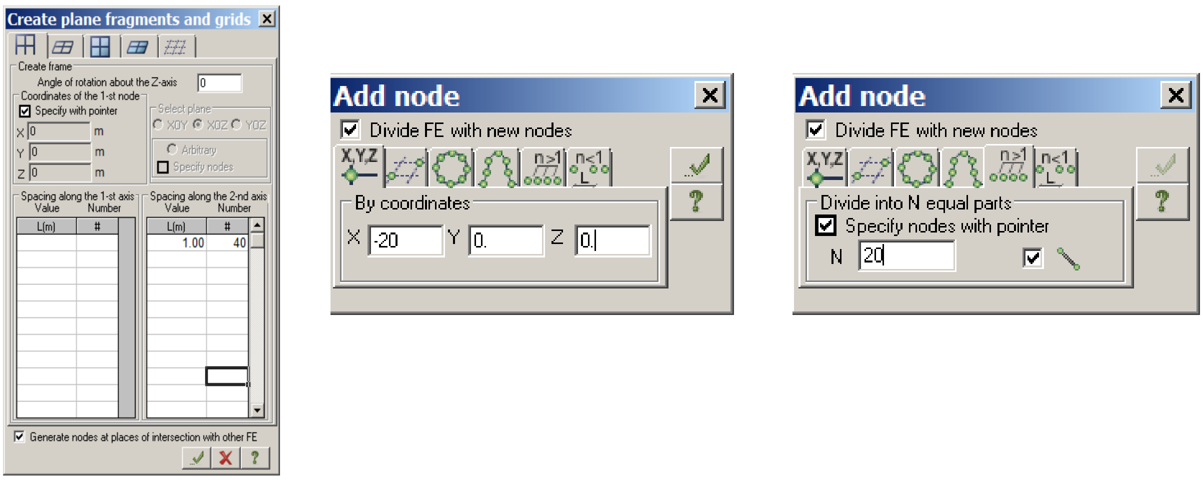


Figure 6.2 – Creation of mast geometric scheme

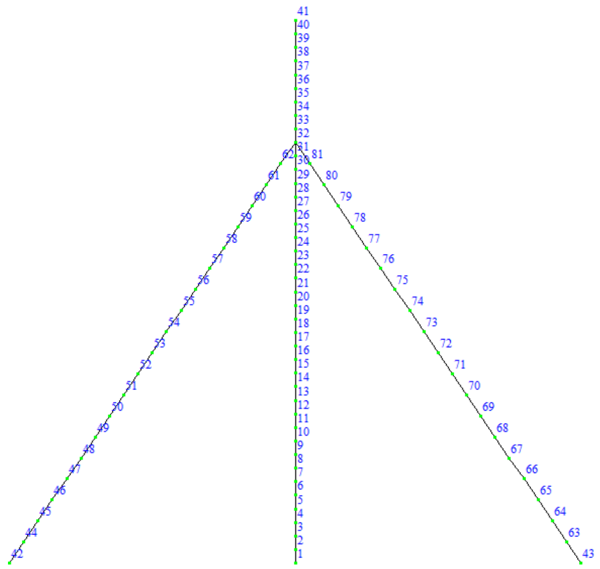


Figure 6.3 – Calculation scheme of the mast

Materials and stiffness

6. Add material “Steel C245” with elasticity modulus 2 × 10⁵ MPa.

7. Assign stiffnesses to the mast bars (tube/angle section according to the standard range).

8. Assign stiffnesses to the stay cables (considering their diameter).

Boundary conditions

9. Restrain the base nodes in X, Y, Z directions.

10. For stays, specify operation in tension only (cable elements).

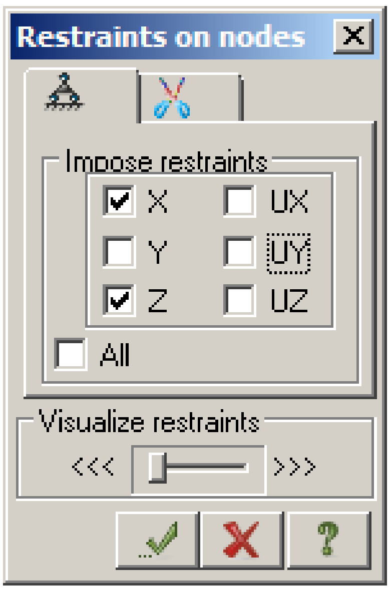


Figure 6.4 – Boundary condition assignment in nodes

Loads

11. Add self-weight.

12. Define wind load: uniform or varying along the height in accordance with standards.

13. Assign initial tensioning of the stays (via initial displacement or applied load).

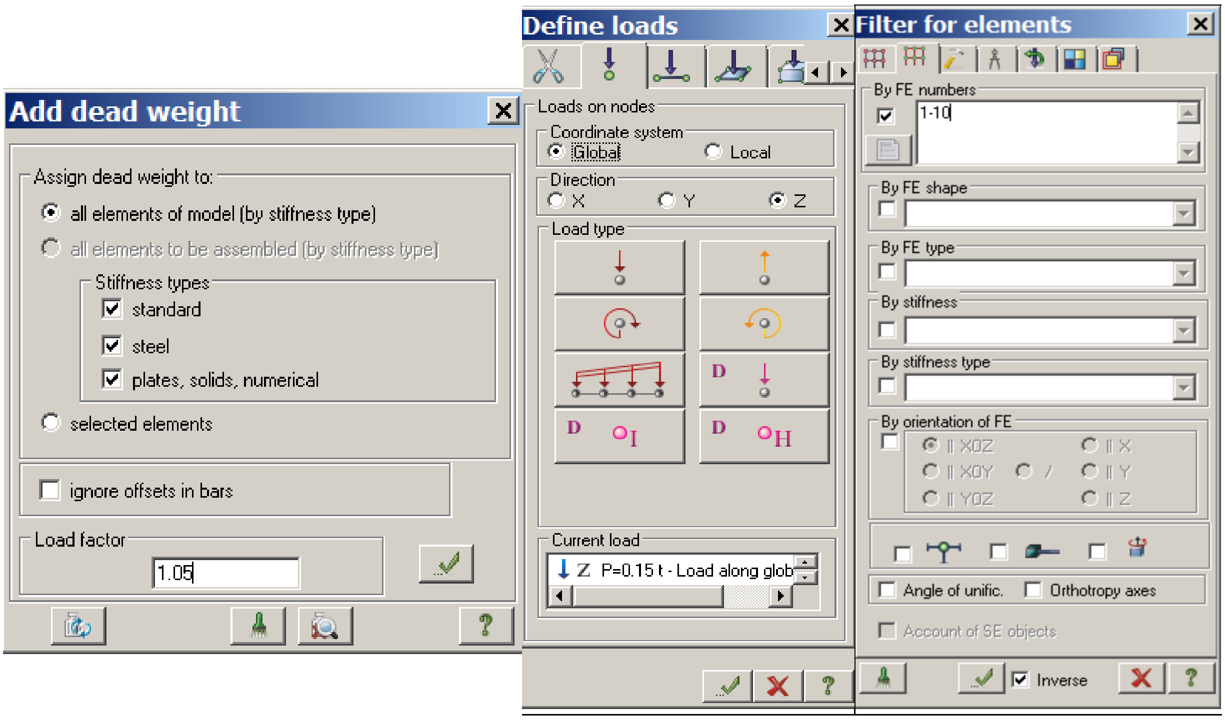


Figure 6.5 – Load assignment

Calculation

14. Perform linear analysis (for comparison).

15. Perform geometrically nonlinear analysis, accounting for shape changes.

Result analysis

16. Compare mast top displacements in linear and nonlinear settings.

17. Review force distribution in the stays and mast bars.

18. Determine the influence of geometric nonlinearity on the calculated stresses.

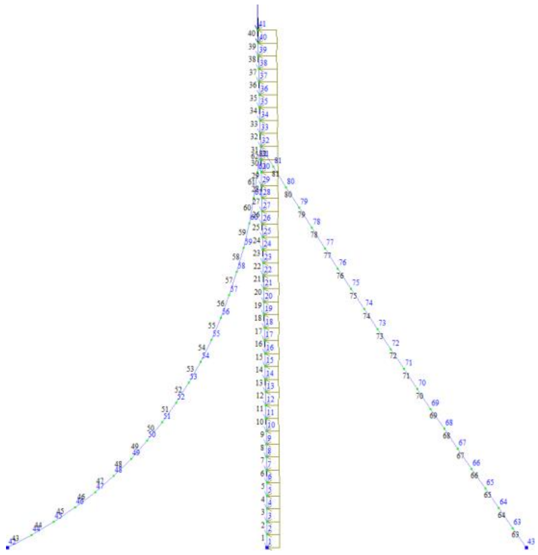


Figure 6.6 – Deformed shape of the structure after nonlinear analysis

Self-assessment questions

1. Why must flexible structures be analysed with consideration of geometric nonlinearity?

2. How does the initial tensioning of stays influence mast stability?

3. What is the difference in cable behaviour between linear and nonlinear settings?

4. How would the analysis change if the stays were modelled as bars working in both compression and tension?

5. Which parameters of the calculation model mesh most significantly influence result accuracy?

PRACTICAL CLASS 7

ANALYSIS OF A SPATIAL FRAME WITH A FOUNDATION SLAB ON AN ELASTIC BASE

The aim of the task is to acquire practical skills in modelling a spatial building frame together with a foundation slab in a software environment. The work involves studying the methodology of accounting for the interaction of the “soil-foundation” system by means of the subgrade modulus for modelling the elastic properties of the soil base.

The exercise includes mastering the use of plate elements for modelling the foundation slab and bar elements for columns and beams of the frame. An important component is the calculation of the spatial model followed by analysis of displacements, internal forces, and stresses in the structural elements.

Initial Data

• Building dimensions: 18 × 12 m, storey height – 3.6 m, number of storeys – 3.

• Material: reinforced concrete C20/25, reinforcement class A500C.

• Subgrade modulus: k = 50,000 kN/m³.

• Supports: foundation slab on an elastic base.

• Loads:

1. Permanent (self-weight of structures).
2. Variable long-term (service loads on floors).
3. Wind load.

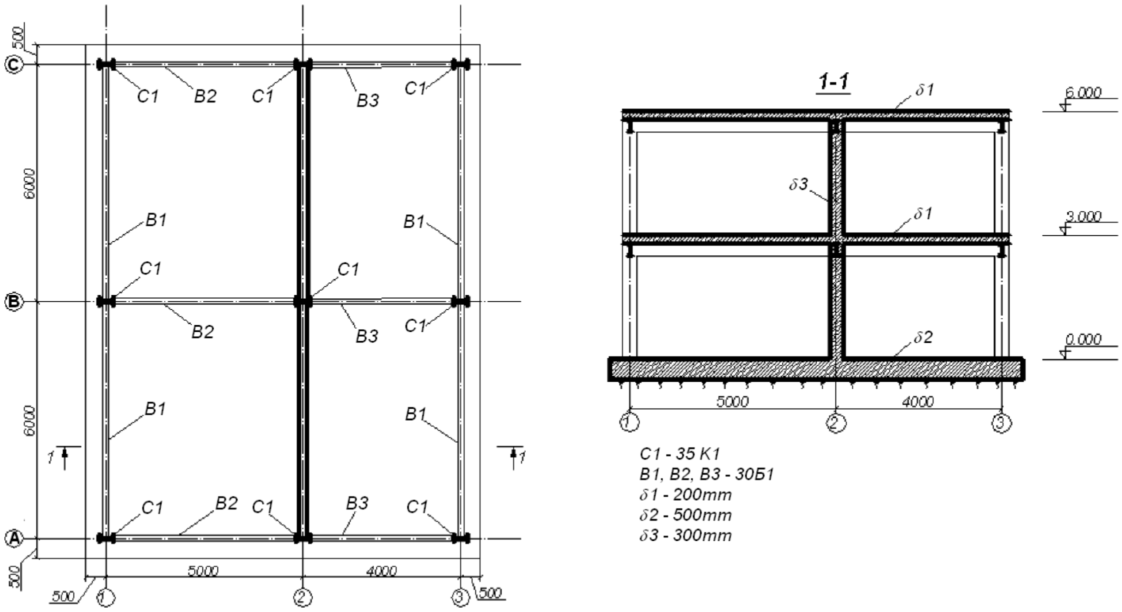


Figure 7.1 – Building frame scheme

Execution Algorithm

Task creation and unit selection

1. Define the system in m, kN.
2. In the settings, enable the use of plate elements for the slab.

Construction of the spatial frame geometry

3. Define the coordinates of the column axes.

4. Model the columns (bar elements) and beams (bar elements at the upper storeys).

5. Create a mesh for the foundation slab (plate elements) with a division of 1–1.5 m.

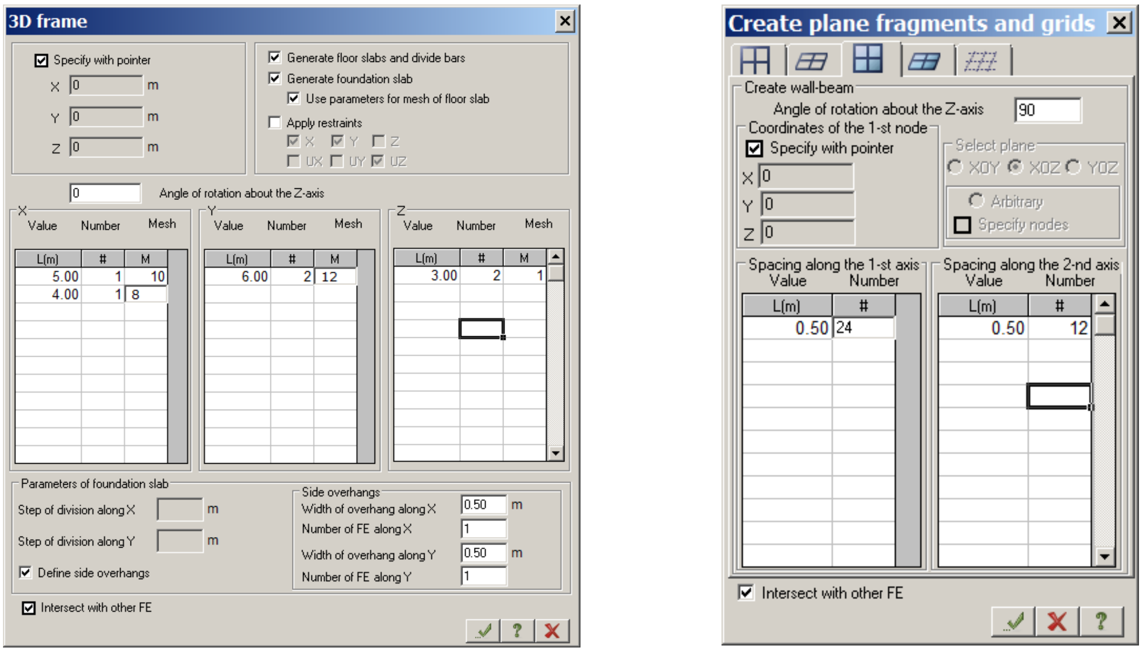


Figure 7.2 – Creation of the geometric scheme

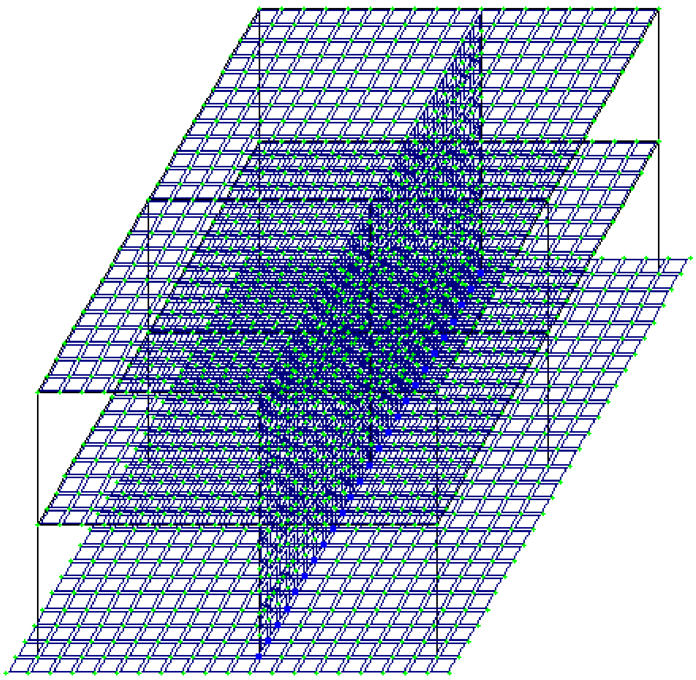


Figure 7.3 – Calculation scheme of the frame

Materials and stiffnesses

6. Add material "RC C20/25" for columns and slab.

7. Assign cross-sections of bars (columns, beams) according to the given dimensions.

8. Specify slab thickness (e.g., 0.4 m).

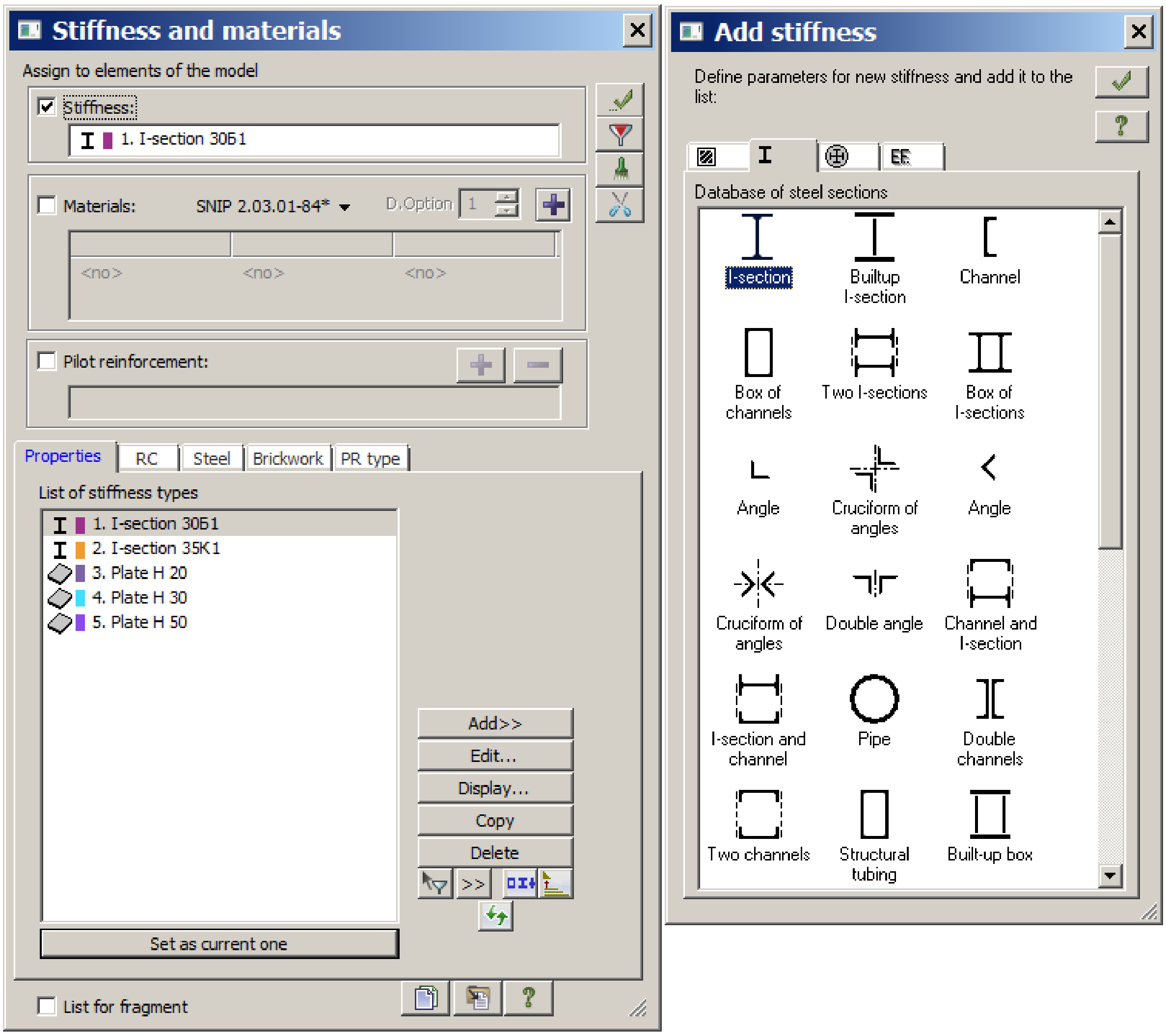


Figure 7.4 – Stiffness parameter assignment

Boundary conditions (elastic base)

9. Assign the subgrade modulus k to the slab nodes.

10. Verify that the restraints account for soil behaviour only in the vertical direction.

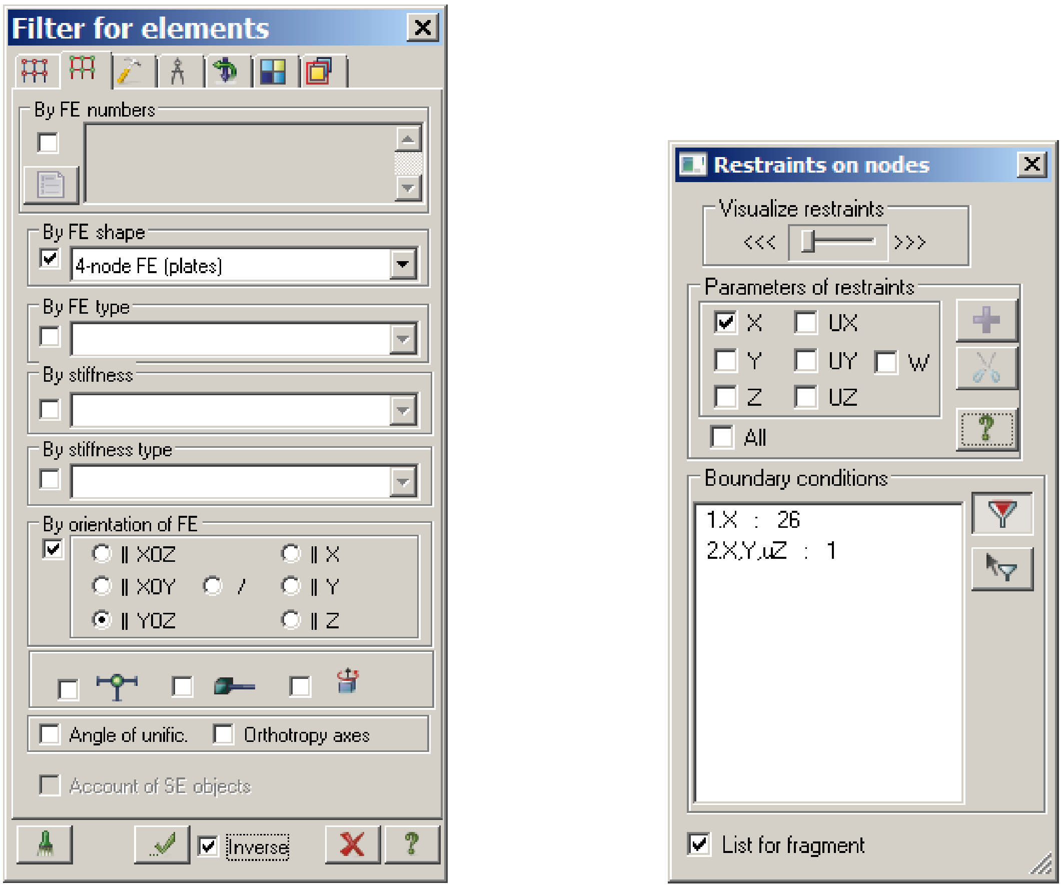


Figure 7.5 – Boundary condition assignment

Load definition

11. Add self-weight.

12. Apply variable loads – on the floors of each storey.

13. Apply wind load – to façade columns and beams with allowance for building height.

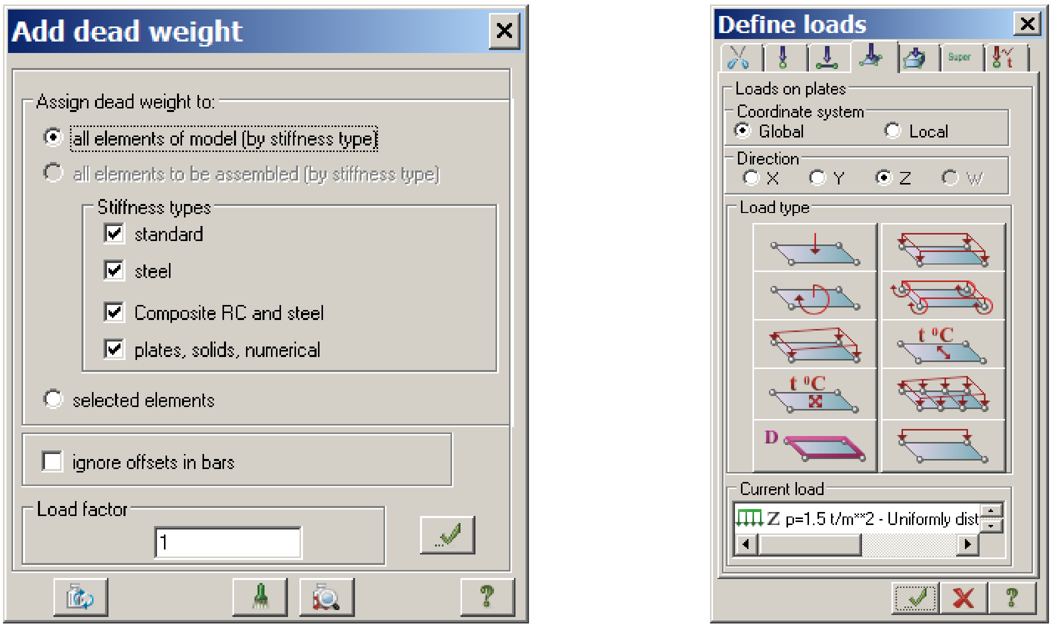


Figure 7.6 – Load assignment

Formation of load combinations

14. Perform automatic generation of load combinations according to DBN or Eurocode.

15. Provide both fundamental and accidental combinations.

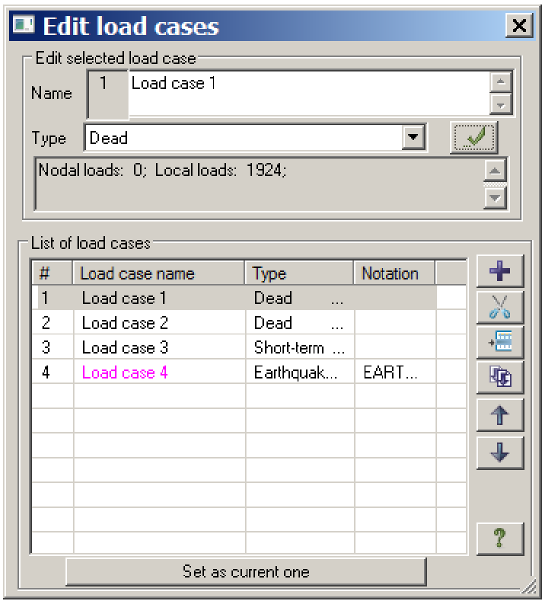


Figure 7.7 – Extended load information assignment

Calculation

16. Perform the analysis of the spatial model.

17. Review displacement maps, internal force diagrams in bars, and Mx, My in the slab.

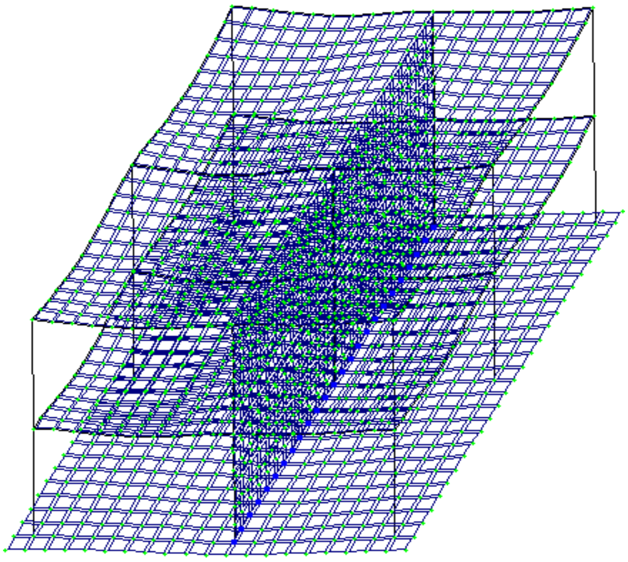


Figure 7.8 – Calculation scheme accounting for node displacements

Result analysis

18. Determine maximum slab deflections and displacements of the top points of the frame.

19. Verify slab stresses and internal forces in column sections.

Self-assessment questions

1. What role does the subgrade modulus play in the analysis of a foundation slab?

2. Why are plate elements used in LIRA-FEM for modelling foundation slabs?

3. How will the stiffness of the system change if slab thickness is increased?

4. What do the M_x and M_y diagrams in the slab represent?

5. How will the analysis change if an absolutely rigid restraint is assumed instead of an elastic base?

PRACTICAL CLASS 8

ANALYSIS OF A SPATIAL FRAME WITH A FOUNDATION SLAB ON A SOIL BASE

The aim of the task is to acquire practical skills in constructing and analysing a spatial model of a building frame with consideration of the interaction of the "soil-foundation" system. The exercise involves mastering the use of an elastic–plastic soil model with the deformation modulus of soil to more accurately reflect the behaviour of the foundation under real loading conditions.

An important component is the calculation of the spatial system with a foundation slab, including the analysis of displacements, internal forces, and stresses in the structural elements, which allows assessment of the stress–strain state of the building as a whole.

Initial Data

• Building dimensions: 24 ×18 m, number of storeys – 5, storey height – 3.3 m.

• Material: reinforced concrete C25/30, reinforcement class A500C.

• Soil deformation modulus: E = 30 MPa (or as specified).

• Foundation slab thickness: 0.5 m.

• Loads: permanent, variable, wind.

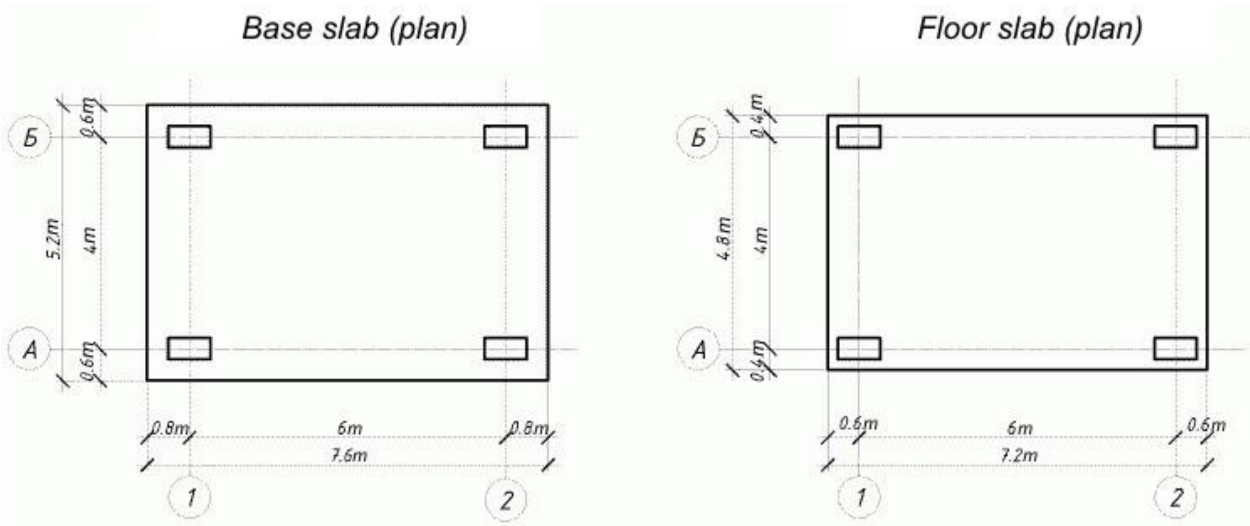


Figure 8.1 – Spatial frame scheme

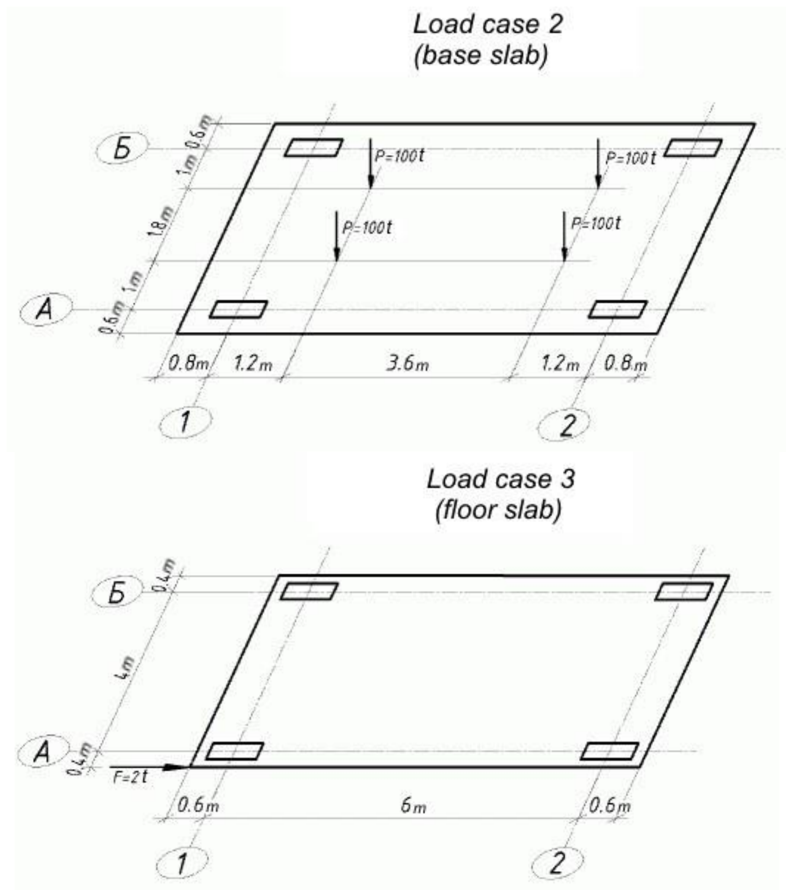


Figure 8.2 – Load schemes of frame slabs

Execution Algorithm

Task creation

1. Select measurement units (m, kN).
2. In the settings, enable the soil base with modulus E.

Model geometry

3. Define the coordinate grid for the columns.
4. Construct the columns, beams, and floor slabs.
5. Create a mesh of the foundation slab with optimal element size (1–1.5 m).

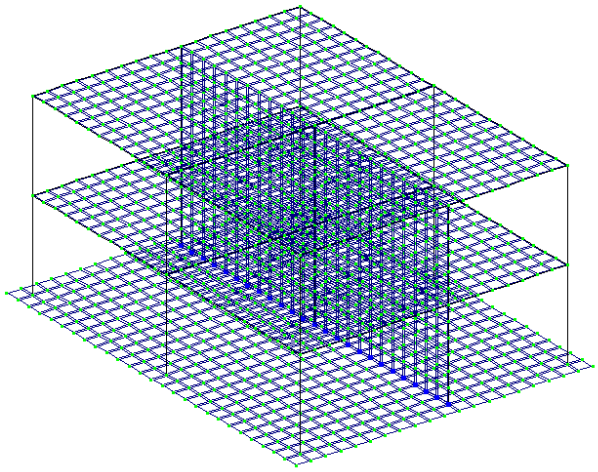


Figure 8.3 – Calculation 3D-model

Materials and stiffnesses

6. Add materials for reinforced concrete and soil (deformation modulus E).

7. Assign bar cross-sections and slab thickness.

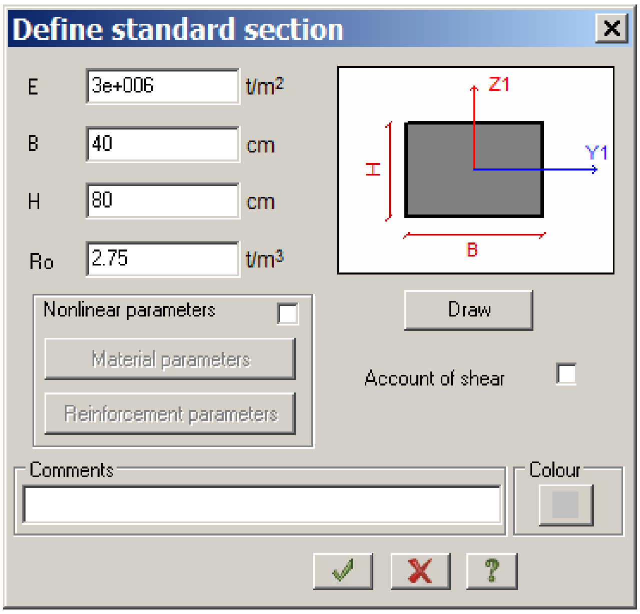


Figure 8.4 – Cross-section assignment

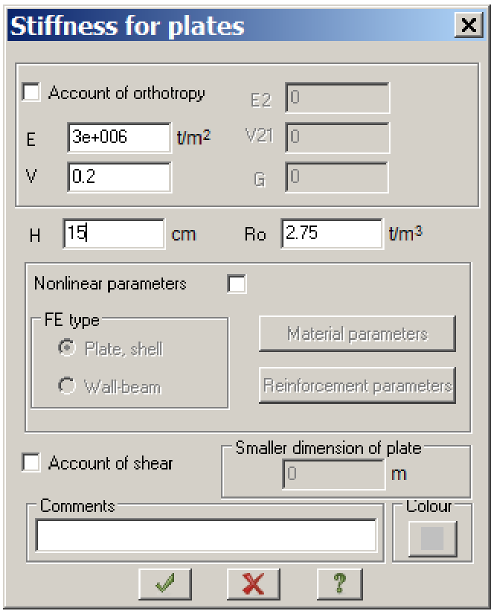


Figure 8.5 – Stiffness assignment

Supports and soil base

8. Assign the soil base to the slab nodes, using modulus E and Poisson's ratio.

9. Verify the working directions of the base (typically vertical and partially horizontal).

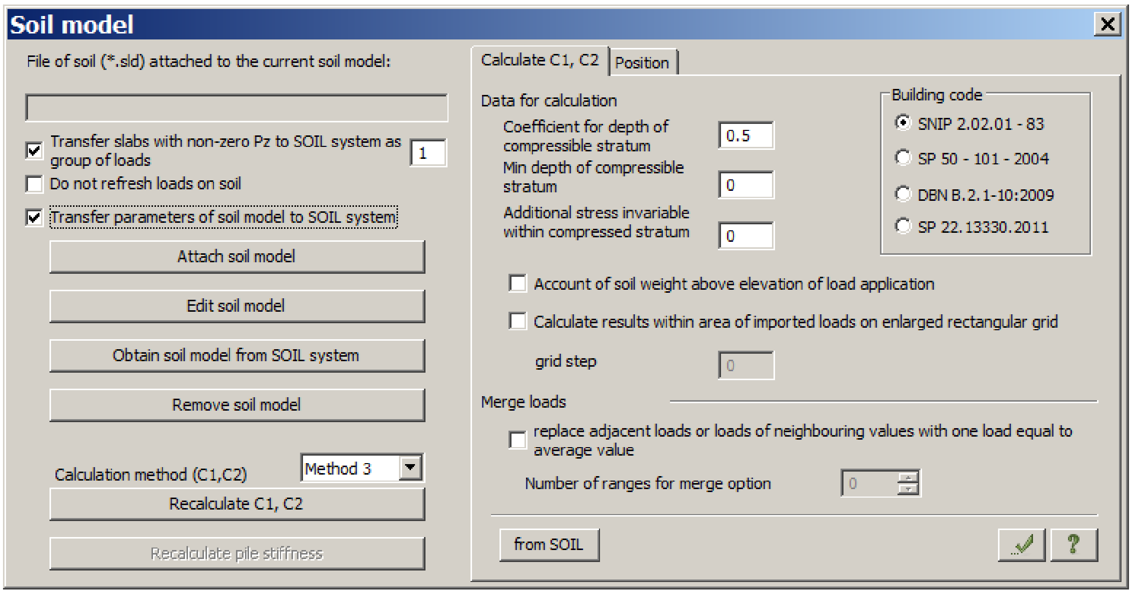


Figure 8.6 – Activation of the “soil” system

Loads

10. Add self-weight of structures.

11. Apply variable loads – on the floor slabs.

12. Apply wind loads – on the external elements of the frame.

Load combinations

13. Generate fundamental and accidental load combinations in accordance with DBN or Eurocode.

Calculation

14. Run the analysis, obtain displacements, internal forces, and moments in the slab and bars.

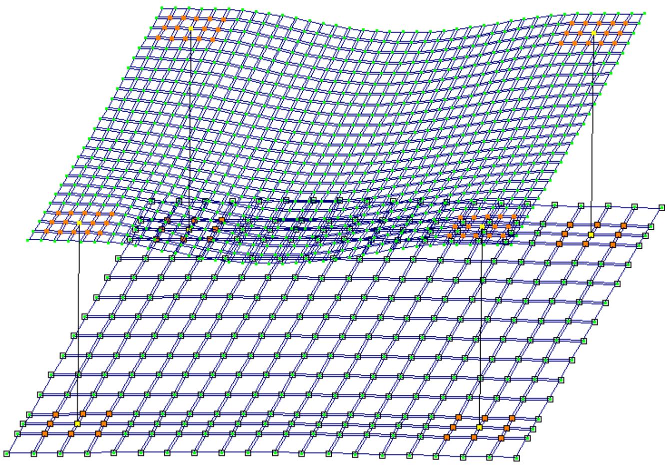


Figure 8.7 – Node displacements

Analysis

15. Assess maximum slab deflections, top displacements of the frame, and distribution of contact pressures under the slab.

16. Verify stresses and forces in the elements.

Self-assessment questions

1. How does a soil base differ from the model with a subgrade modulus?

2. How does the soil deformation modulus affect system displacements?

3. Why is Poisson's ratio of soil required in calculations?

4. How will the results change if the slab thickness is increased?

5. Why is it important to analyse the distribution of contact pressures beneath the slab?

PRACTICAL CLASS 9

ANALYSIS OF A PHYSICALLY NONLINEAR BEAM WITH CONSIDERATION OF CONCRETE CREEP

The aim of the task is to become familiar with methods of modelling physical nonlinearity of reinforced concrete elements in the LIRA-FEM software package. The exercise involves learning to account for concrete creep when performing engineering calculations and analysing the effect of long-term loads on the stress–strain state of a beam, in particular on the magnitude of its deflections.

An additional objective is to consolidate skills in defining material parameters using "σ–ε" diagrams for concrete and steel, enabling a more accurate representation of the actual structural behaviour under service conditions.

Initial Data

1. The beam scheme and its supports are shown in figure 9.1.

2. Beam cross-sections are shown in figure 9.2.

3. Beam material – reinforced concrete C20/25, reinforcement A400C.

4. The calculation model is analysed after 365 and 730 days.

5. Loads:

• Permanent (self-weight + structural load).

• Long-term variable (service load).

6. Calculation conditions: consideration of physical nonlinearity and concrete creep.

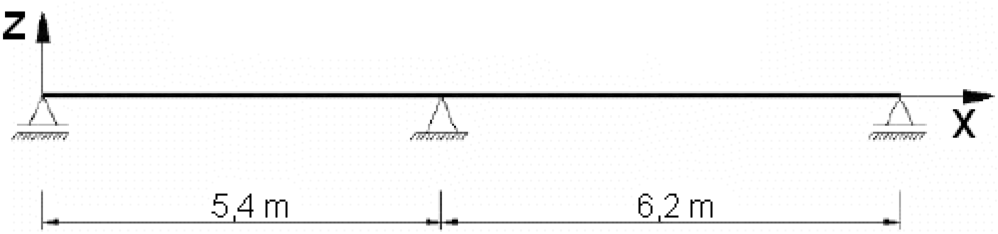


Figure 9.1 – Beam scheme

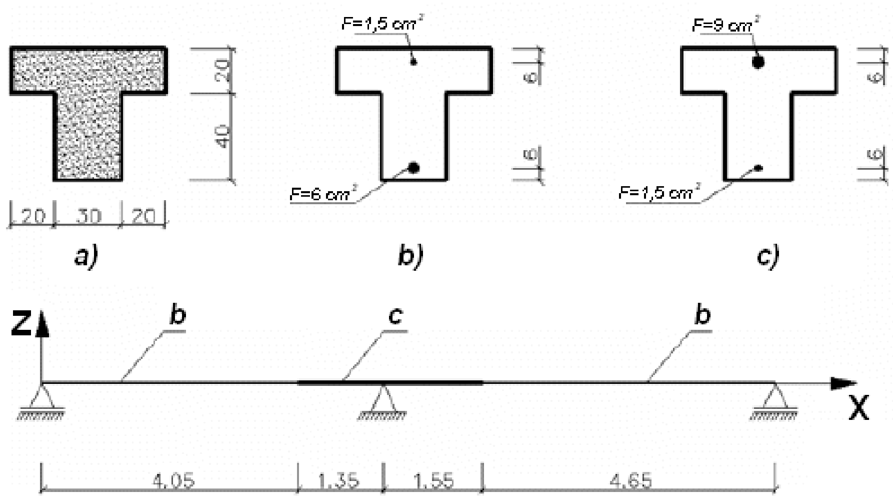


Figure 9.2 – Beam cross-sections

Execution Algorithm

Creating a new task

1. Launch LIRA-FEM, create a new project file.

2. Set the system of measurement units: metres, kN.

3. Activate the option for physically nonlinear analysis in the project settings.

Geometry definition

4. Create nodes in plan according to the span lengths.

5. Connect the nodes with bar elements to form the geometric beam model.

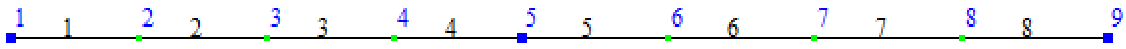


Figure 9.3 – Analytical beam scheme

Cross-section definition

6. Select the cross-section type as rectangular reinforced concrete.

7. Specify width b and height h according to the initial data.

8. Verify that the section complies with the concrete strength class.

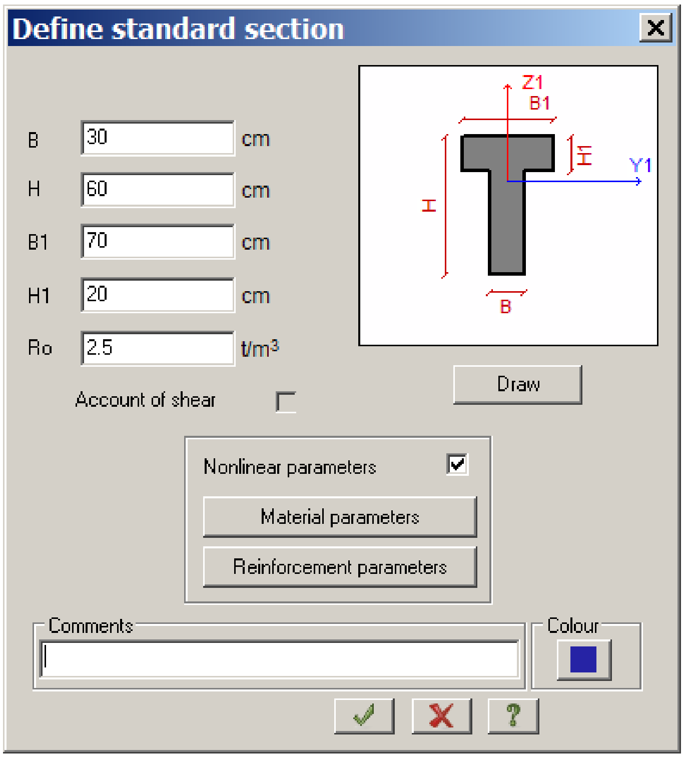


Figure 9.4 – Beam cross-section definition

Material definition

9. Create a new material: concrete C20/25:

• Assign the relevant “σ–ε” deformation diagram for PNA analysis.

• Define modulus of elasticity *E*, Poisson’s ratio *ν*, and ultimate strains.

10. Create a material for reinforcement A400C with its own “σ–ε” diagram.

11. Assign reinforcement to the beam cross-section (longitudinal and transverse reinforcement as specified).

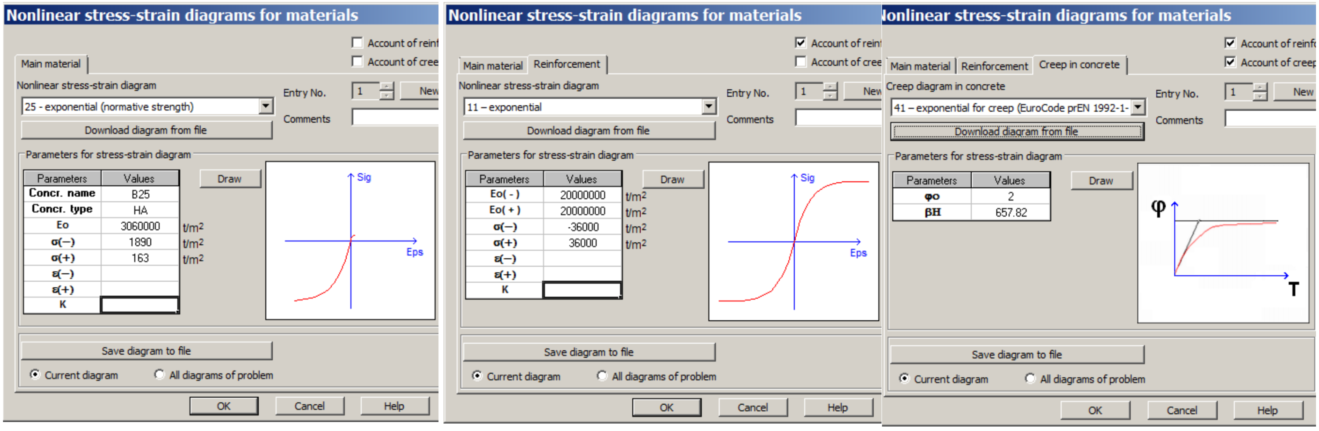


Figure 9.5 – Definition of nonlinear deformation laws of materials

Concrete creep consideration

12. In the concrete material properties, open the "Creep and Shrinkage" section.

13. Enable creep consideration.

14. Input parameters: creep coefficient φ(t, t_0) or creep modulus according to DBN/Eurocode 2.

15. Specify the load duration.

16. If required by the code, account for concrete shrinkage.

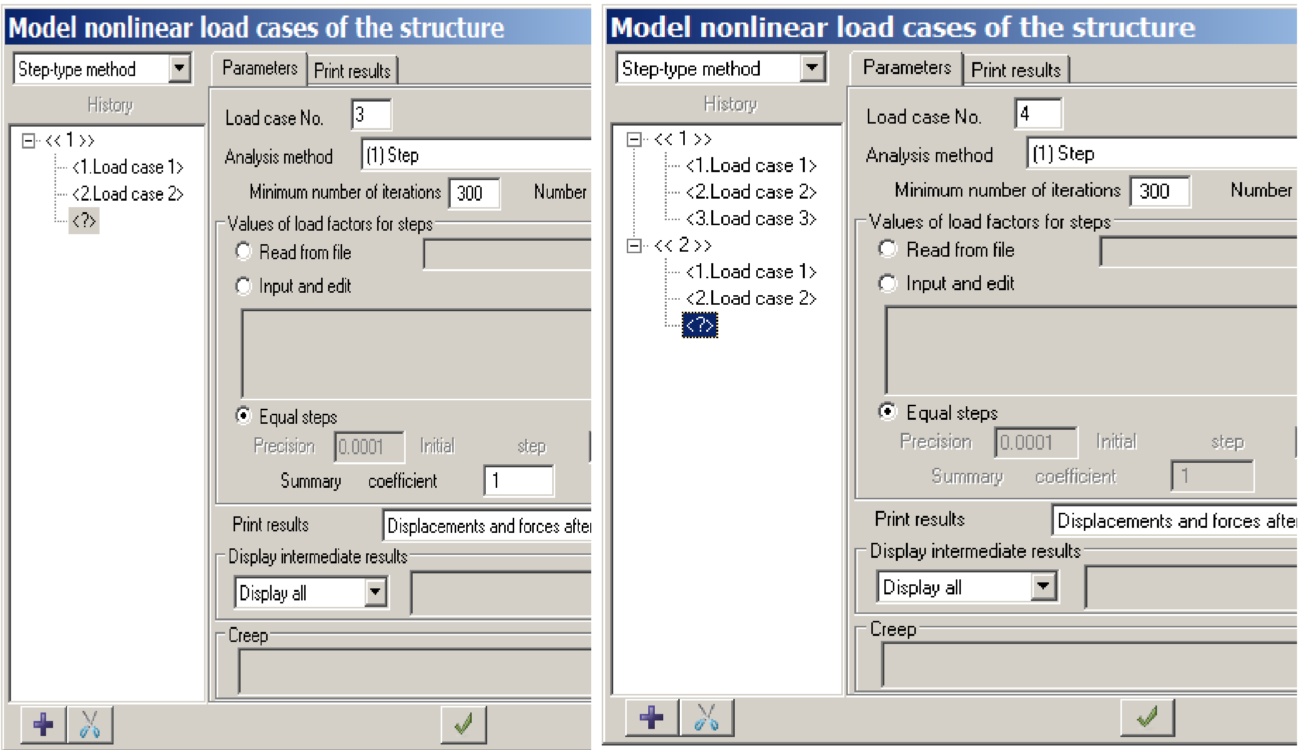


Figure 9.6 – Modelling nonlinear loading

Load definition

17. Self-weight: enable automatic calculation of structural self-weight.

18. Permanent load: apply uniformly distributed load along the beam length (e.g., from floor structures, finishes, etc.).

19. Long-term variable load: define as a separate load case.

20. Assign load groups (Permanent, Long-term Variable) for correct combinations.

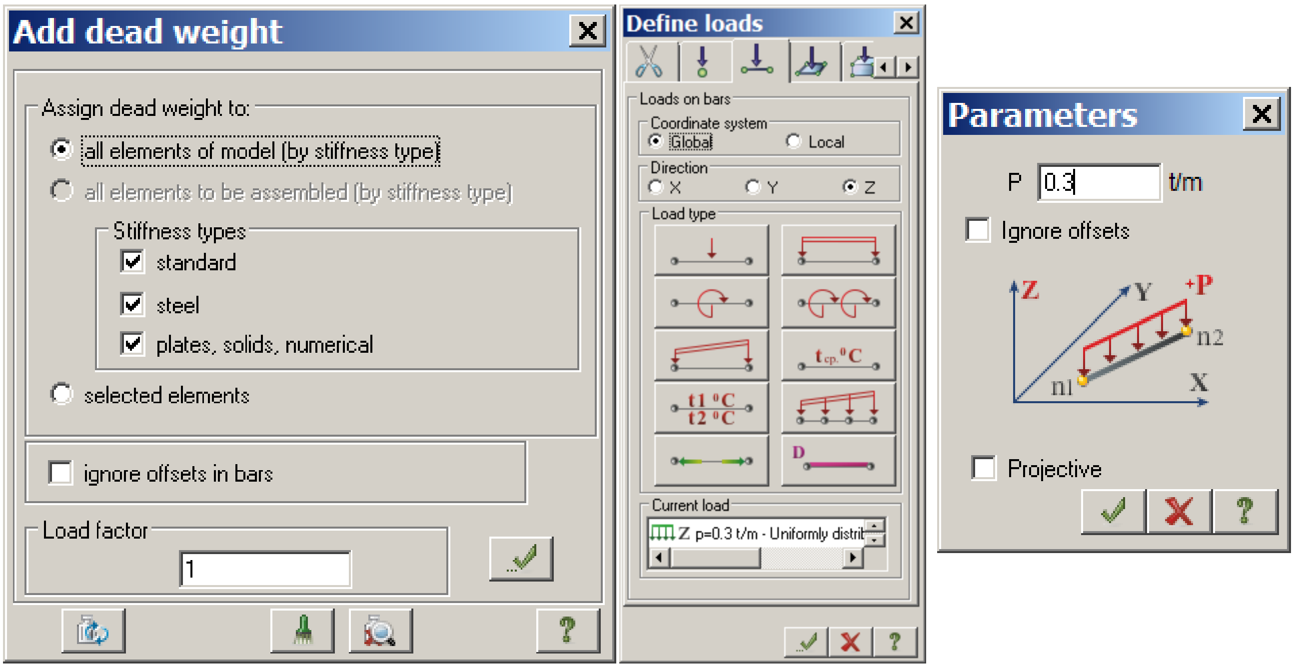


Figure 9.7 – Load definition

Boundary conditions

21. For a simply supported beam, restrain the support nodes against vertical displacements (and horizontal if required).

22. Ensure there are no kinematic indeterminacies in the scheme. *Calculation settings*

23. In the calculation dialogue, selectphysically nonlinear analysis with creep consideration.

24. Define load steps (e.g., 10–20 steps for smooth diagram generation).

25. Activate the option for long-term load analysis.

Performing the calculation

26. Run the analysis.

27. Verify the calculation protocol for errors or warnings.

Results analysis

28. Review displacement maps – determine maximum deflections.

29. Review stress maps in concrete and reinforcement.

30. Analyse the “moment–curvature” diagram for the beam.

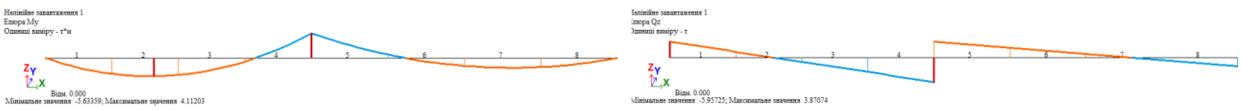


Figure 9.8 – Internal force diagrams

31. Compare results with analysis excluding creep consideration:

• How have deflections changed?

• How have stresses in concrete and reinforcement changed?

32. Draw conclusions on the influence of long-term loads.

Self-assessment questions

1. What is physical nonlinearity and how is it modelled in LIRA-FEM?

2. How are concrete creep parameters determined by design codes?

3. How does creep differ from shrinkage?

4. What is the effect of long-term loads on beam deflections and stresses?

5. Why is it important to perform comparative analysis “with” and “without” creep consideration?

6. How does the stress distribution in concrete change under long-term loading?

PRACTICAL CLASS 10

ANALYSIS OF A SHEET PILE WALL REINFORCED WITH ANCHORS IN INTERACTION WITH A SOIL MASS (LIRA-FEM)

The aim of this exercise is to acquire practical skills in modelling and analysing retaining structures of an excavation, represented as a sheet pile wall reinforced with an anchoring system interacting with a multilayered soil base. Within this objective, the task involves practising the application of a physically nonlinear soil model according to the Coulomb–Mohr criterion, which enables the realistic simulation of soil mass behaviour under loads and changing construction conditions.

Another objective is to develop an understanding of staged construction analysis, which reflects the actual sequence of wall installation and excavation works. In addition, the task includes studying the influence of anchor pre-tensioning on reducing sheet pile wall displacements and redistributing forces within the "soil–structure" system. Achieving this objective provides a comprehensive understanding of the behaviour of retaining structures in complex geotechnical conditions and lays the foundation for further engineering calculations related to the design of excavation supports and underground structures.

Initial Data

1. Base geometry: profile 60 × 20 m, layer thickness 1 m; mesh element spacing 1 × 1 m. Excavation: 20 × 8 m. Retaining wall: vertical elements 14 m high. Anchor system: supports (forcops) 3 m long; anchors 10 m long at an inclination of 45°.

2. Soil layer materials (Coulomb–Mohr model, FE 281–284):

• Layer 1 (fill): E = 800 t/m²; ν = 0.3; h = 100 cm; γ = 1.6 t/m³; c = 0.1 t/m²; Rt = 0.01 t/m²; φ = 30°; transition coefficient to secondary branch Ke = 3.

• Layer 2 (sand):* E = 3 000 t/m²; ν = 0.3; h = 100 cm; γ = 1.7 t/m³; c = 0.1 t/m²; Rt = 0.01 t/m²; φ = 30°.

Note. Y-axis – perpendicular to the plane; use the scheme option “2 – three DOF (X, Z, U_y).

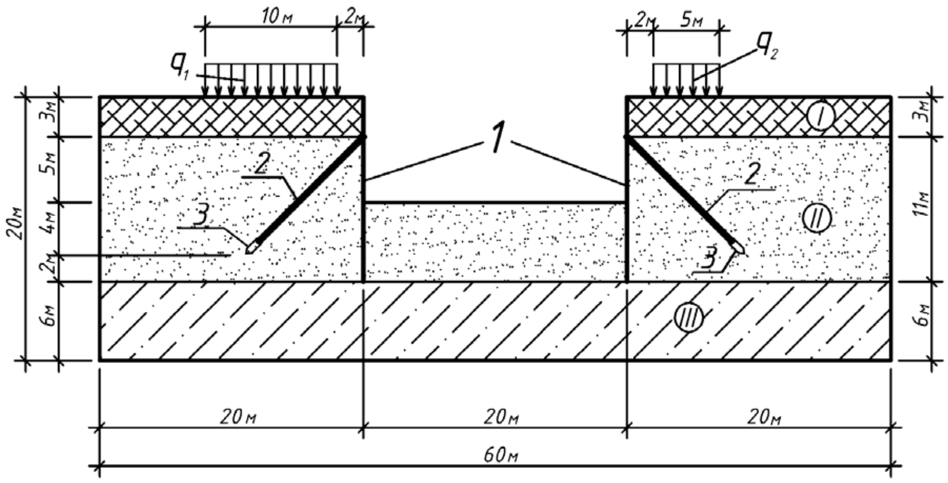
3. Loads:

• Self-weight of the three-layer soil mass.

• Permanent uniformly distributed loads on the soil surface: g_1 = 1 t/m; g_2 = 0.5 t/m; + self-weight of retaining wall elements.

• Anchor pre-tensioning F = 5 t (special element).

• Fictitious point load P = 0.25 t applied to the edge point of the soil base in the direction of boundary conditions.



1 - sheet piling
2 - anchors
3 - supports for anchors
I - fill-up soil
II - sand
III - clay loam

Figure 10.1 – Scheme of retaining wall structures of the excavation and loads on the multilayer base

Execution Algorithm

Task creation

1. Name: 10_sheet_pile_with_soil_mass.

2. Plane: X0Z, “2 – three DOF (X, Z, U_y)”.

Mesh and finite element type

3. Create a rectangular area 60 × 20 m with 1 m spacing.

4. Assign type 281 (physically nonlinear rectangular finite element for plane soil analysis) to the area.

5. Enable node numbering display if required.

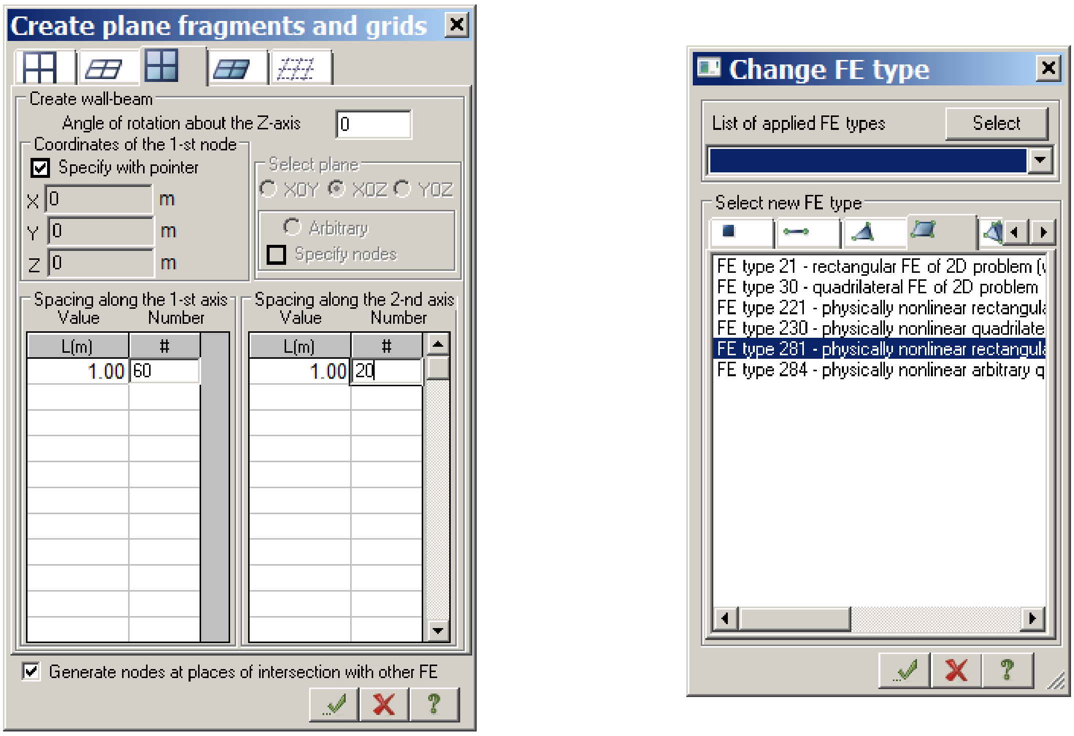


Figure 10.2 – Creating the geometric scheme

Insertion of retaining wall and anchors

6. Add sheet pile wall bars (height 14 m) along excavation boundaries.

7. Add anchor supports (≈3 m) and anchors 10 m long at 45°.

8. Ensure correct connectivity of wall nodes with the soil mesh.

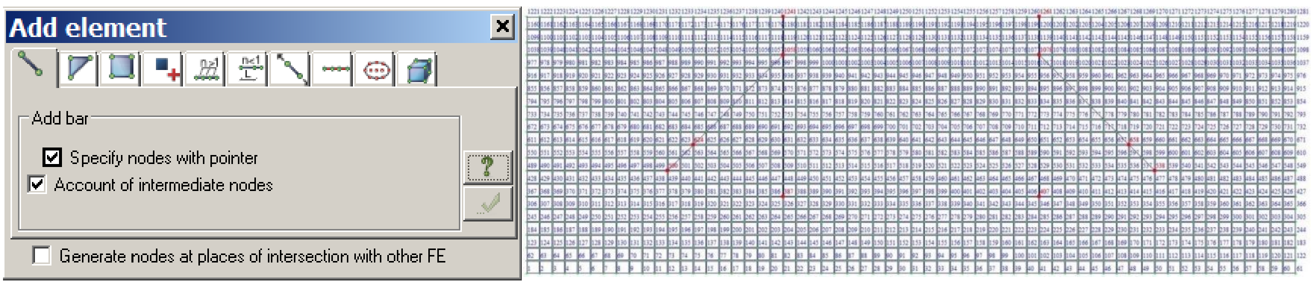


Figure 10.3 – Inserting retaining wall and anchors

Boundary conditions

9. At the bottom boundary of the base, restrain X and Z displacements.

10. At the side boundaries – apply plane-strain boundary conditions (restraining normal displacements).

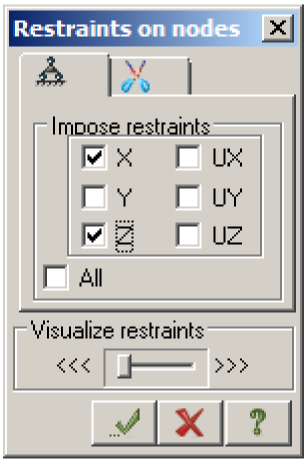


Figure 10.4 – Defining boundary conditions

Soil materials and stiffness

11. Create finite element 281–284 (numerical description) for Layer 1 (fill) with the parameters above.

12. Copy the stiffness type twice for Layer 2 and Layer 3; edit the parameters.

13. Assign colours/comments for convenient layer identification.

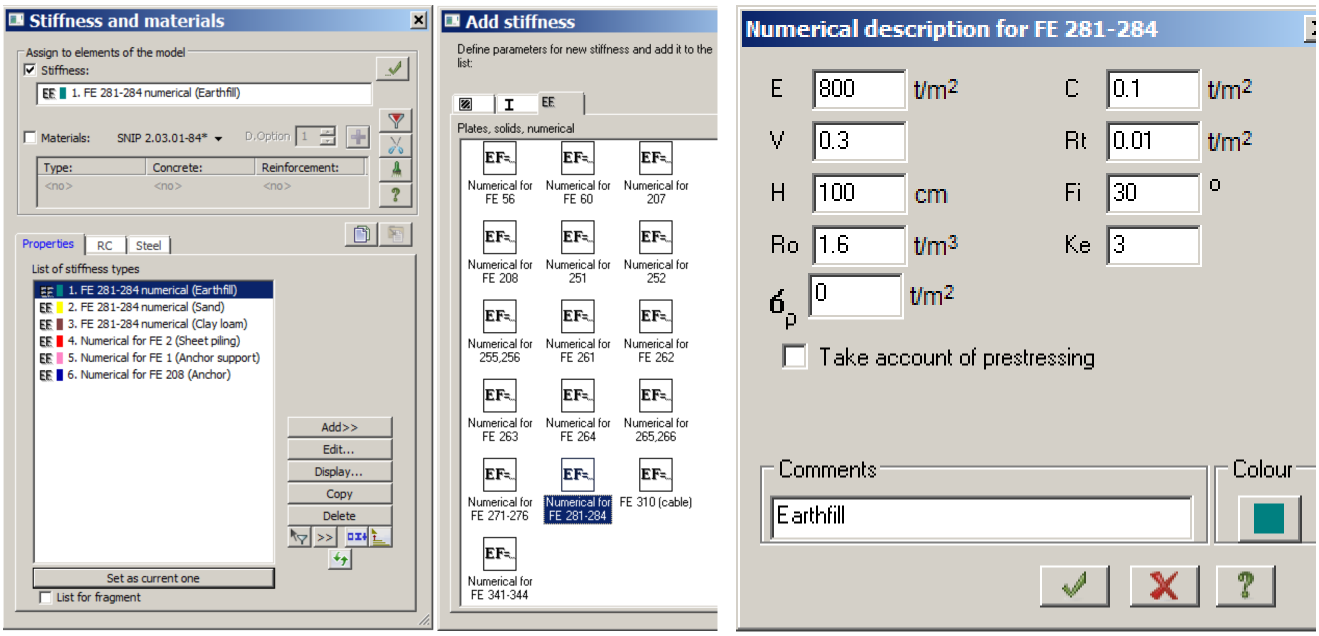


Figure 10.5 – Assigning stiffness parameters to elements

Load definition

14. Add self-weight.

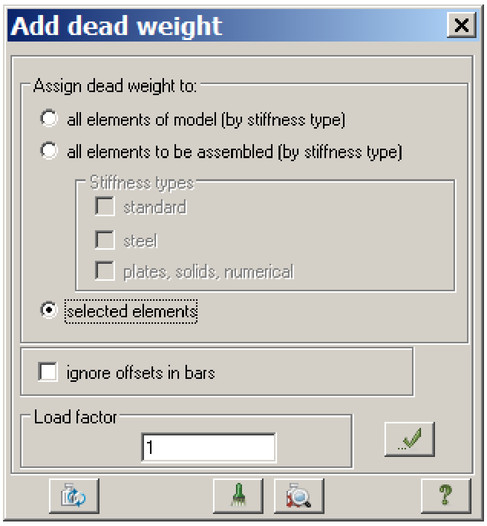


Figure 10.6 – Self-weight definition

15. Apply g_1 = 1 t/m, g_2 = 0.5 t/m on the surface + self-weight of the wall.

16. In the polyfilter, select Type 208 (special two-node pre-tension element). On the “Bar Loads” tab, define the forcops with P = 5 t.

17. Apply point load P = 0.25 t at the extreme node of the base in the direction of boundary conditions.

Staged construction and nonlinear loading

18. Stage 1:base – self-weight.

19. Stage 2:wall installation (stiffness “4. Finite element 2 numerical (Wall)” → “All selected”). In the second nonlinear loading case, check the option “Reset displacements”.

20. Stage 3:excavation of the 1st layer inside the retaining wall (select “1. FE 281–284 (Fill)” within the excavation).

21. Stage 4:installation/activation of anchors and pre-tensioning (reference load case №4).

22. Stage 5:completion, fictitious load for correct sequencing.

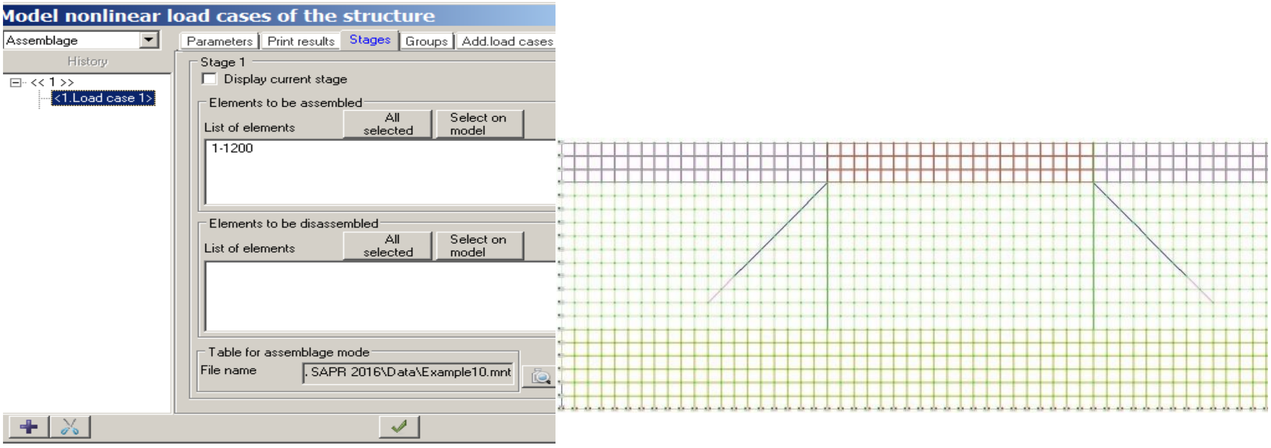


Figure 10.7 – Load definition

Calculation

• Run a physically nonlinear analysis with Coulomb–Mohr model and transition to the secondary branch (Ke).

• Obtain displacements, internal forces, and moments in the wall and soil elements.

• Assess maximum deflections.

• Check stresses and forces in elements.

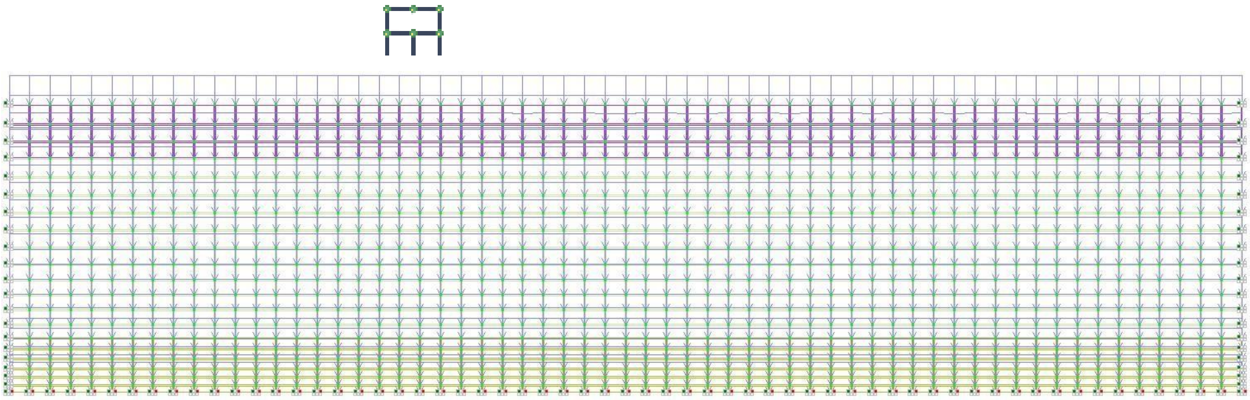


Figure 10.8 – Analytical scheme considering node displacements

Self-assessment questions

1. What is the role of “resetting displacements” in Stage 2?

2. How do the primary and secondary soil deformation branches differ, and what is the influence of the Ke coefficient?

3. How does anchor pre-tensioning affect the behaviour of the sheet pile wall and the soil mass?

4. How would the results change with a different anchor inclination or length?

PRACTICAL CLASS 11

ANALYSIS OF A STEEL BUILDING FRAME WITH PREPARATION OF DATA FOR PROGRESSIVE COLLAPSE ASSESSMENT

The aim of this task is to master the methodology of modelling a steel frame of a multi-storey building in the LIRA-FEM software package. The assignment involves learning how to prepare input data for subsequent progressive collapse analysis, which is an important stage in assessing the reliability and safety of a building.

An essential component is the development of skills in creating spatial models of steel frames and verifying their load-bearing capacity in accordance with current design standards, which enables a comprehensive evaluation of the frame performance under both service and accidental load conditions.

Initial Data

• The frame scheme and its supports are shown in figure 11.1.

• Cross-sections of elements:

1. Edge and middle columns – I-beam № 35 K1.
2. Longitudinal beams – I-beam № 30.
3. Transverse beams – welded I-beam.
4. Bracing of columns – two angles 75 × 75 × 6.

• Structural system: steel braced frame.

• Materials: steel grade S355 for main elements.

• Loads: permanent, variable, wind.

• Software: LIRA-FEM, 3D analysis mode.

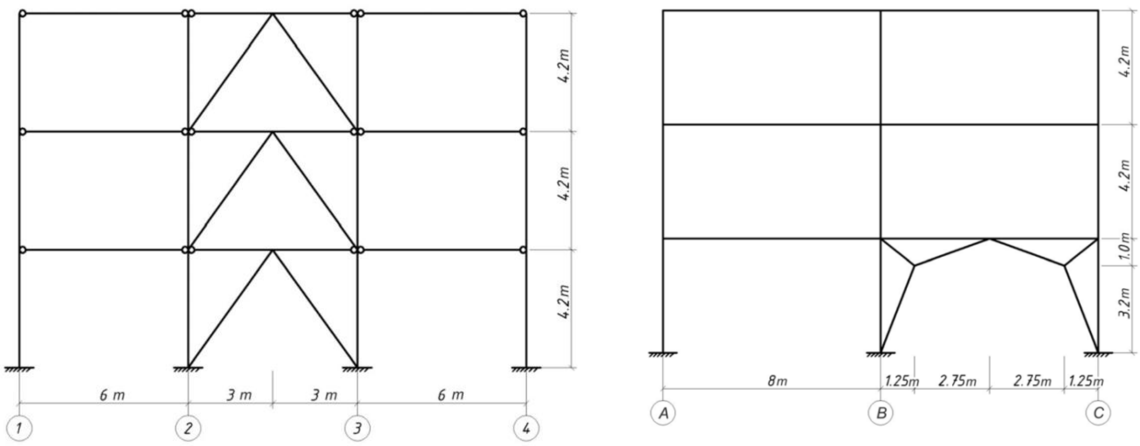


Figure 11.1 – Analytical scheme of the building frame cross-section

Execution Algorithm

Creating a new task

1. Launch LIRA-FEM, set measurement units – metres, kN.

2. Create a new project file, enable 3D analysis mode.

Constructing the geometric scheme

3. Define node coordinates according to the building grid and storey heights.

4. Connect nodes with bar elements (columns, beams, bracing).

5. Check node connectivity (no gaps in the model).

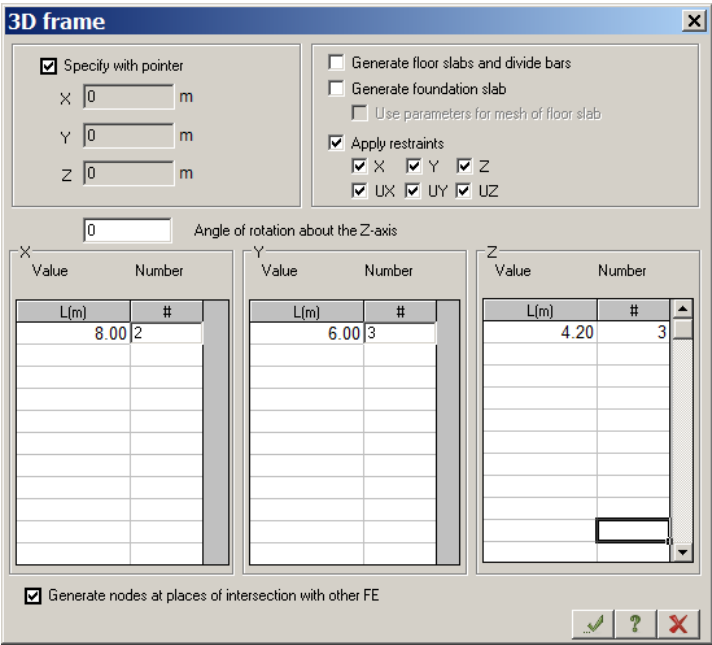


Figure 11.2 – Constructing a spatial frame

Assignment of cross-sections and materials

6. Assign hot-rolled I-sections or built-up sections to columns according to the catalogue.

7. Assign I-sections or channels to beams according to the variant.

8. For vertical and horizontal bracing, assign circular or angle profiles.

9. Assign material: steel S355.

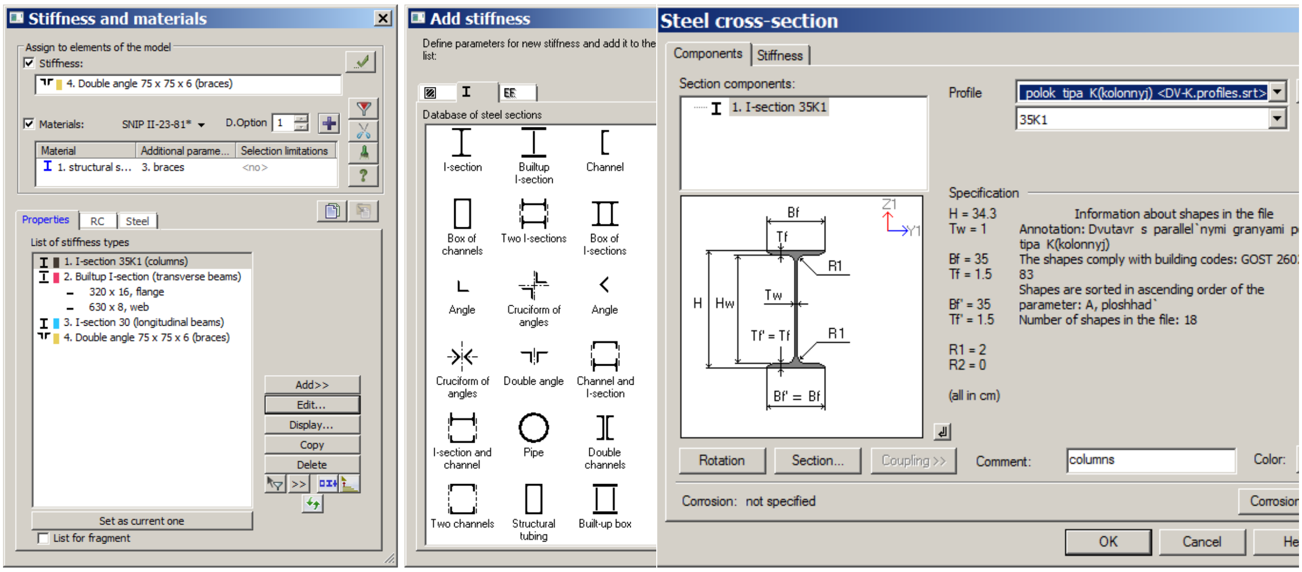


Figure 11.3 – Assignment of stiffness characteristics and material properties to elements

Boundary conditions

10. Column supports at ground floor – pinned or rigidly fixed (as specified).

11. At roof level nodes, if required, assign elastic restraints.

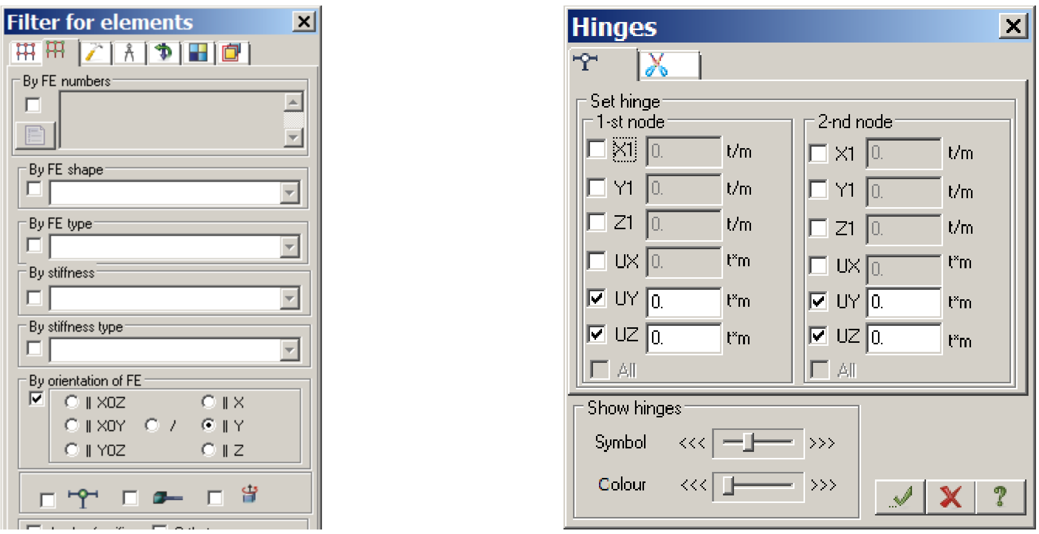


Figure 11.4 – Defining boundary conditions

Load definition

12. Permanent loads: self-weight of structures, weight of enclosing structures.

13. Variable loads: imposed, snow, wind (in accordance with DBN or Eurocode).

14. Apply loads at relevant nodes or directly to elements.

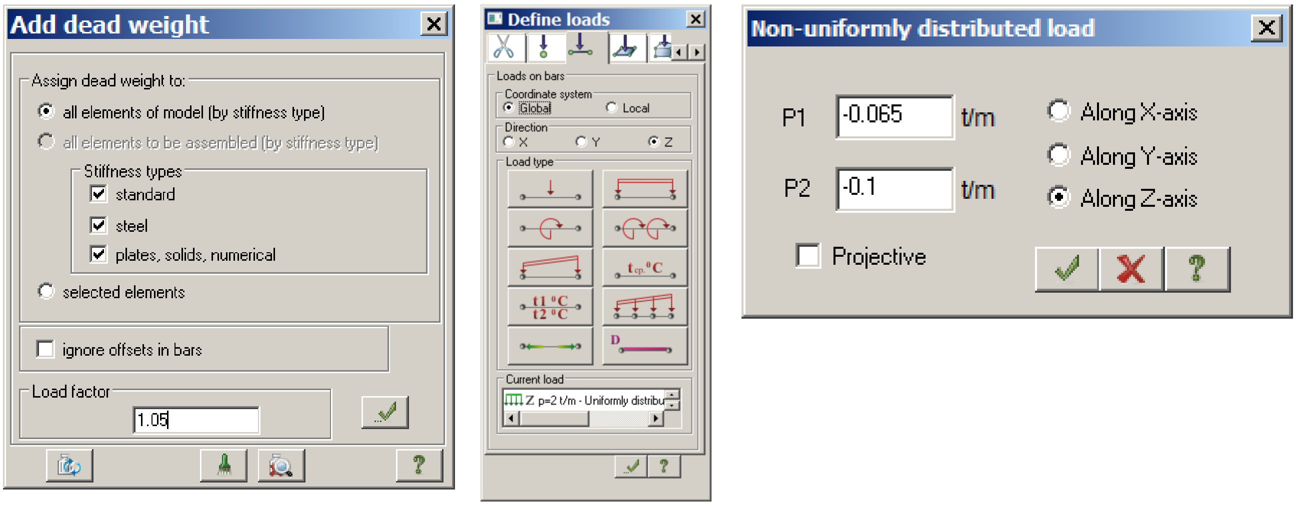


Figure 11.5 – Load definition

Formation of load combinations (LC)

15. Generate LC for ultimate and serviceability limit states (ULS, SLS).

16. Add combinations to be used in modelling accidental scenarios (element removal).

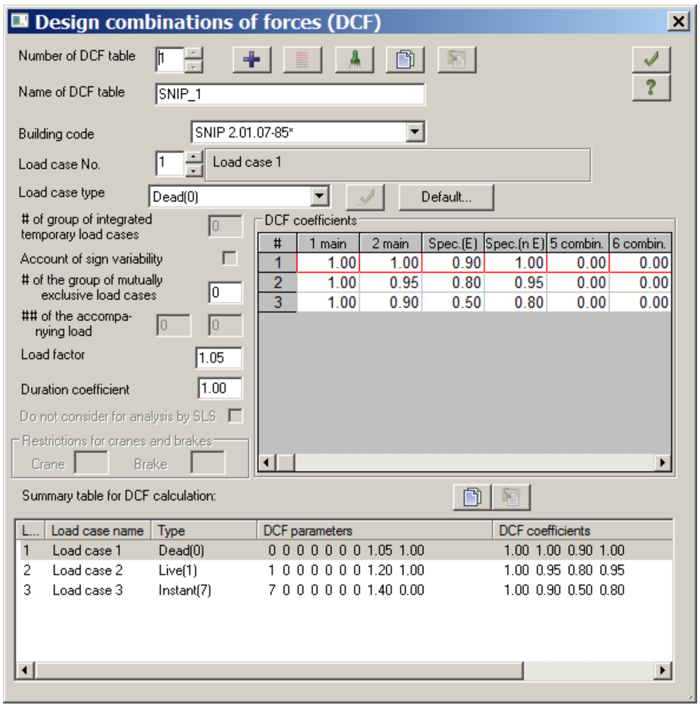


Figure 11.6 – Load combination table

Frame analysis

17. Run linear static analysis.

18. Review displacements and internal forces, check absence of instability.

19. If necessary, adjust cross-sections to ensure strength.

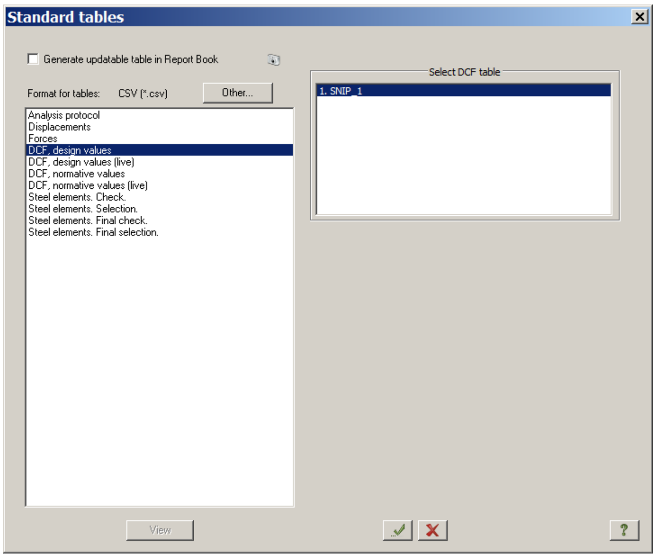


Figure 11.7 – Reviewing calculation results tables

Preparation for progressive collapse analysis

20. Save the basic model as a separate file.

21. In the copy, remove specific elements (columns, beams) according to the failure scenario.

22. Check the new scheme for stability and redistribution of forces.
Result analysis

23. Identify maximum displacements and stresses in the damaged scheme.

24. Compare with permissible values.

25. Formulate conclusions on the reserve of load-bearing capacity.

Self-assessment questions

1. What are the specific features of modelling steel frames in LIRA-FEM?

2. How do bracing members affect the frame stiffness?

3. What is the methodology of preparing a model for progressive collapse analysis?

4. Which load combinations should be applied for accidental load cases?

5. How should the results of analysis be interpreted when elements are removed?

PRACTICAL CLASS 12

ANALYSIS OF A SPATIAL REINFORCED CONCRETE FRAME IN A PHYSICALLY NONLINEAR FORMULATION

The purpose of this work is to master the methodology of modelling spatial reinforced concrete frames in a physically nonlinear formulation using the LIRA - FEM software package. The task involves learning the principles of working with nonlinear material models that more accurately represent the real physical – mechanical properties of concrete and reinforcement.

Special attention is paid to the consideration of cracking processes and the redistribution of internal forces within the structure, which ensures higher accuracy in predicting its stress–strain state and assessing load-bearing capacity under service loads.

Initial Data

• Two-span, three-storey building.

• Span length – 6 m, column spacing – 6 m, storey height – 3 m.

• Columns at the foundation slab level – rigidly fixed. Material: reinforced concrete C20/25, reinforcement A400C.

• Cross-sections of elements:

1. Columns of the first and last frames – rectangular section 500 × 500 mm.

2. Edge columns of the central frame – T-section, height 1 200 mm (flange width – 1 200 mm, flange thickness – 300 mm, web thickness – 300 mm).

3. Central columns of the middle frame – I-section, height 600 mm (flange width – 600 mm, flange thickness – 200 mm, web thickness – 200 mm).

4. Floor and roof beams – rectangular section 400 × 500 mm.

5. Floor and roof slabs – thickness 200 mm.

• Structural elements: columns, beams, slabs.

• Material: concrete class C32/40, reinforcement class A500C.

• Loads: permanent, variable, snow, wind.

• Calculation scheme: spatial.

Execution Algorithm

Creating a new task

1. Select units – metres, kN.

2. Create a spatial analysis problem.

Geometric model

3. Input node coordinates according to the building grid and storey heights.

4. Define bar elements (columns, beams) and plate elements (slabs).

5. Check model integrity (absence of "hanging" nodes).

Assignment of materials and cross-sections

6. Assign a physically nonlinear concrete model (e.g., Concrete 55 in LIRA-FEM).

7. For reinforcement, assign a model with a stress–strain diagram.

8. Assign cross-sections to elements as per the specification.

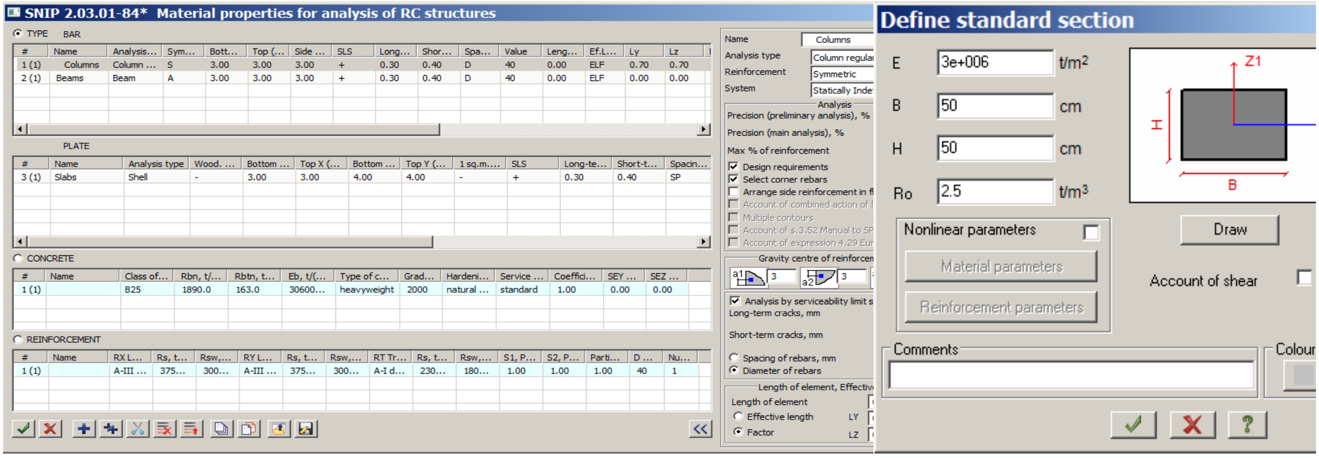


Figure 12.1 – Assignment of materials and cross-sections

Boundary conditions

9. Fix the nodes at the base of the columns (rigid fixity or hinge).

10. If necessary, define elastic supports simulating soil-structure interaction.

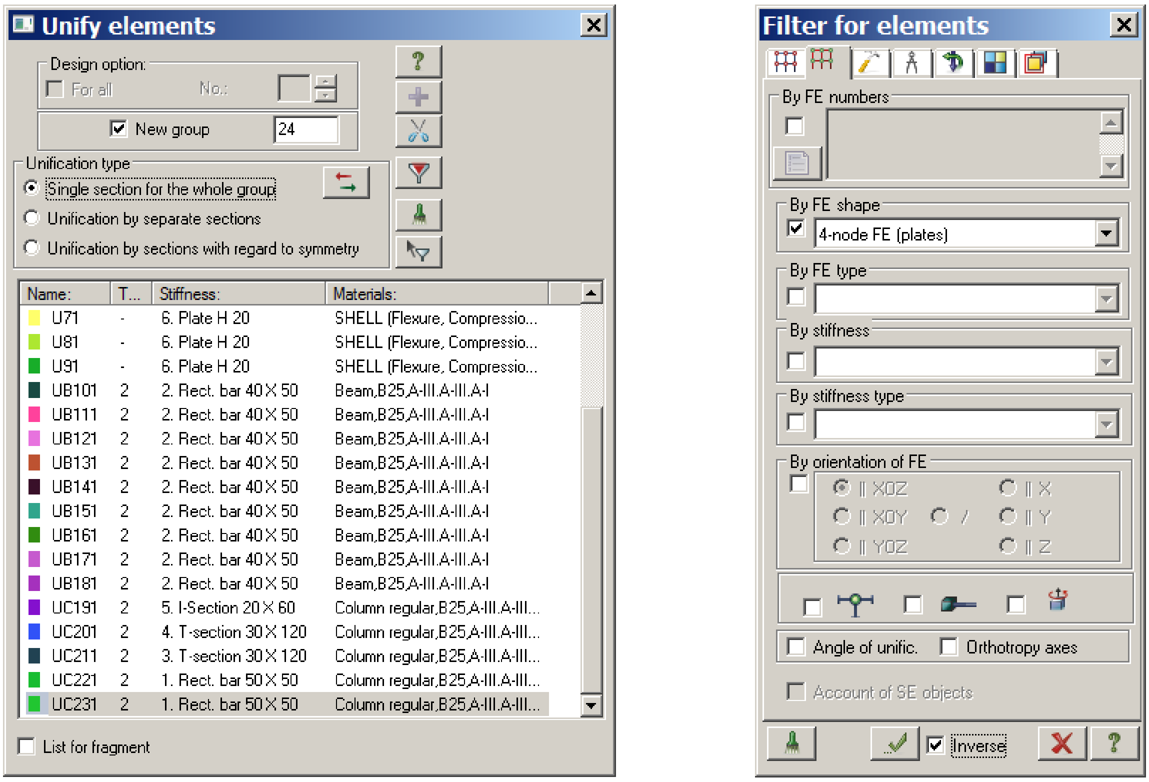


Figure 12.2 – Element unification

Load definition

11. Permanent: self-weight of structures, partitions, roofing.

12. Variable: imposed (service) loads.

13. Climatic: snow and wind.

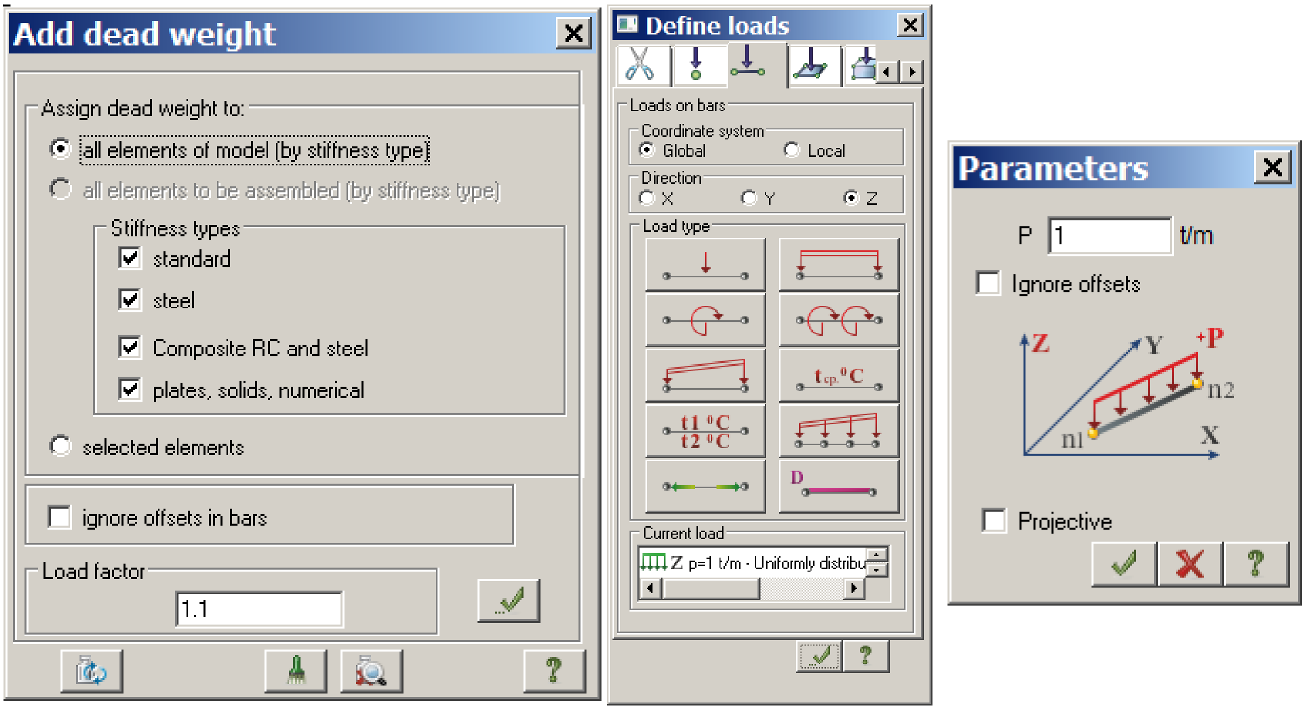


Figure 12.3 – Load definition

Formation of load combinations

14. ULS – ultimate limit states.

15. SLS – serviceability limit states.

Settings for physically nonlinear analysis

16. Select nonlinearity type – physical.

17. Define parameters of material stress-strain diagrams.

18. Enable cracking analysis for reinforced concrete elements.

Execution of analysis

19. Run physically nonlinear analysis.

20. Check the results of step-by-step loading.

21. Assess zones of plastic deformations and crack widths.

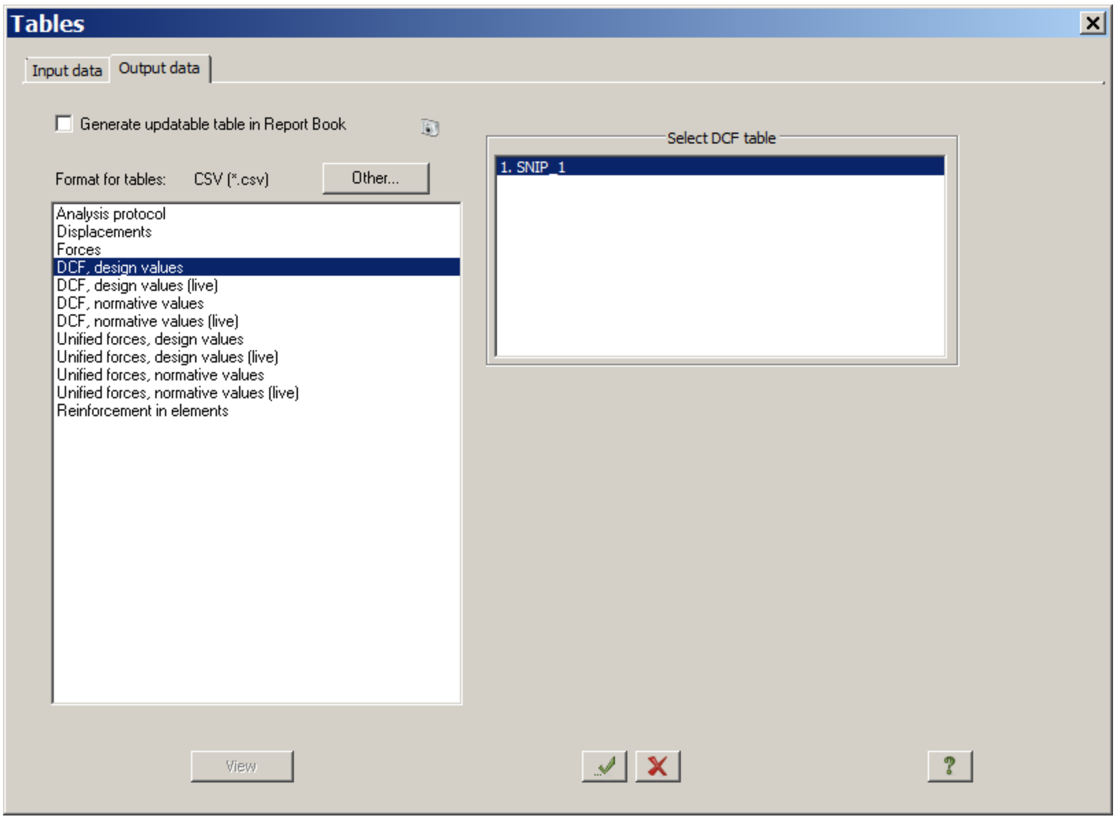


Figure 12.4 – Review of result tables

Analysis of results

22. Determine maximum deflections and displacements.

23. Review stress maps in concrete and reinforcement.

24. Compare with permissible limits (DBN, Eurocode).

25. Formulate conclusions regarding load-bearing capacity.

Self-assessment questions

1. What is the difference between physical and geometric nonlinearity?

2. How is concrete behaviour considered after crack formation?

3. Why is step-by-step loading important in nonlinear analysis?

4. How are zones of plastic deformations identified in the model?

5. Which key parameters must be defined for the reinforcement model?

PRACTICAL CLASS 13

ANALYSIS OF A STEEL TRUSS WITH CIRCULAR SECTIONS

The purpose of this work is to demonstrate the procedure for analysing a joint of a steel truss composed of circular tubular members using modern calculation methods. The task involves illustrating the procedure for editing user-defined parameters and verifying the load-bearing capacity of the joint in accordance with regulatory requirements.

An additional objective is to familiarise students with the specifics of modifying the geometric and stiffness characteristics of structural members in order to ensure the required strength and reliability of the joint under design loads.

Initial Data

- Truss geometry and supports – see figure 13.1.
- Assumed member cross-sections:
  1. Top chord – seamless hot-rolled tube Ø159 × 7.5 mm.
  2. Bottom chord – seamless hot-rolled tube Ø133 × 5 mm.
  3. End diagonals (1–10, 2–10, 9–17, 8–17) – tube Ø76 × 5 mm.
  4. Other diagonals – tube Ø63.5 × 5 mm.
- Material: steel.
- Coefficients:
  5. Service condition factor $\gamma_c = 0.9$.
  6. Reliability factor by purpose $\gamma_n = 0.95$.
- Design forces in the joint – see figure 20.2:
  7. Chord: $N_x = –255$ kN.
  8. Diagonal 1: $N_x = 120$ kN.
  9. Diagonal 2: $N_x = –196$ kN.

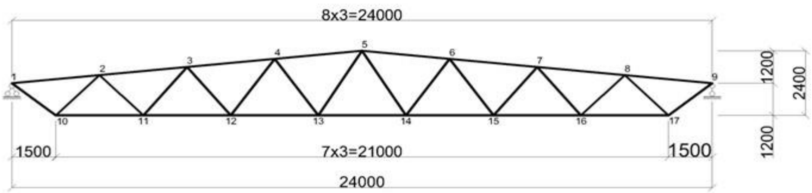


Figure 13.1 – Geometry of the steel truss

Execution Algorithm

Launching and creating a new joint

1. Create a new joint: File → New → Joint, or use the toolbar button.

2. In the Select Joint Type window, choose:

• Structure type: Truss joints.

• Subtype: Circular tube (select the appropriate option from the illustration).

3. Click OK.

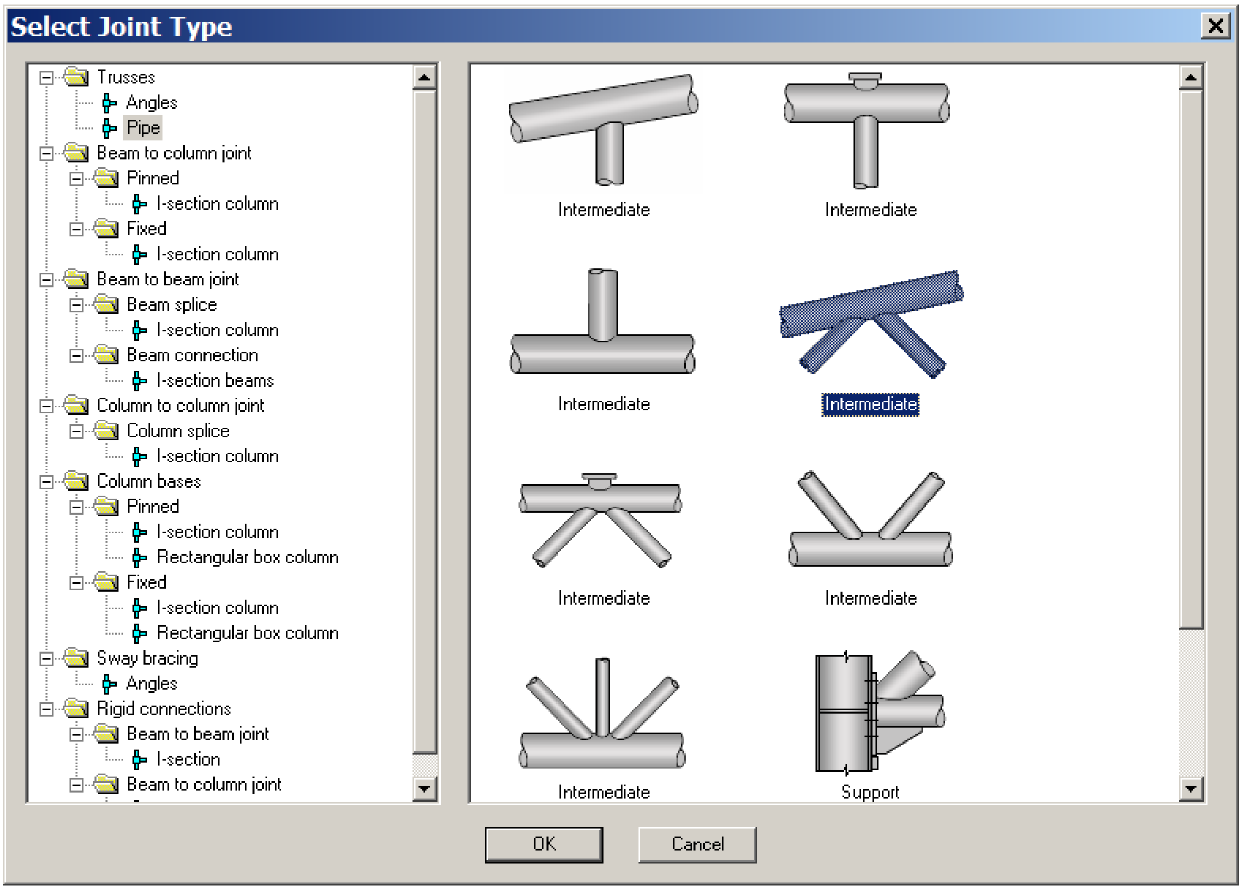


Figure 13.2 – Selection of joint type

General joint parameters

4. In the General dialogue window specify:

• Joint name – Truss Joint 1.

• $\gamma_n = 0.95$.

• $\gamma_c = 0.9$.

5. For the Welds group:

• Properties: service condition factor $\gamma_c = 0.9$; wire grade – Sv-10GA.

6. If required, configure Risk Factors:

• Standardised value: Z25.

• Joint stiffness: Medium.

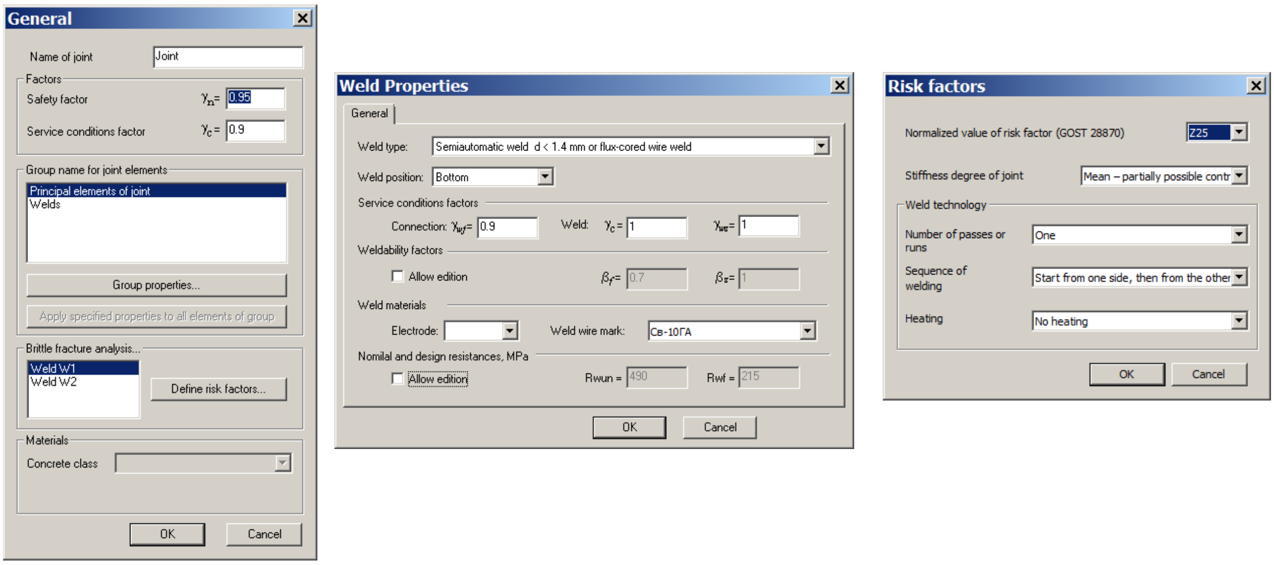


Figure 13.3 – Assignment of joint parameters

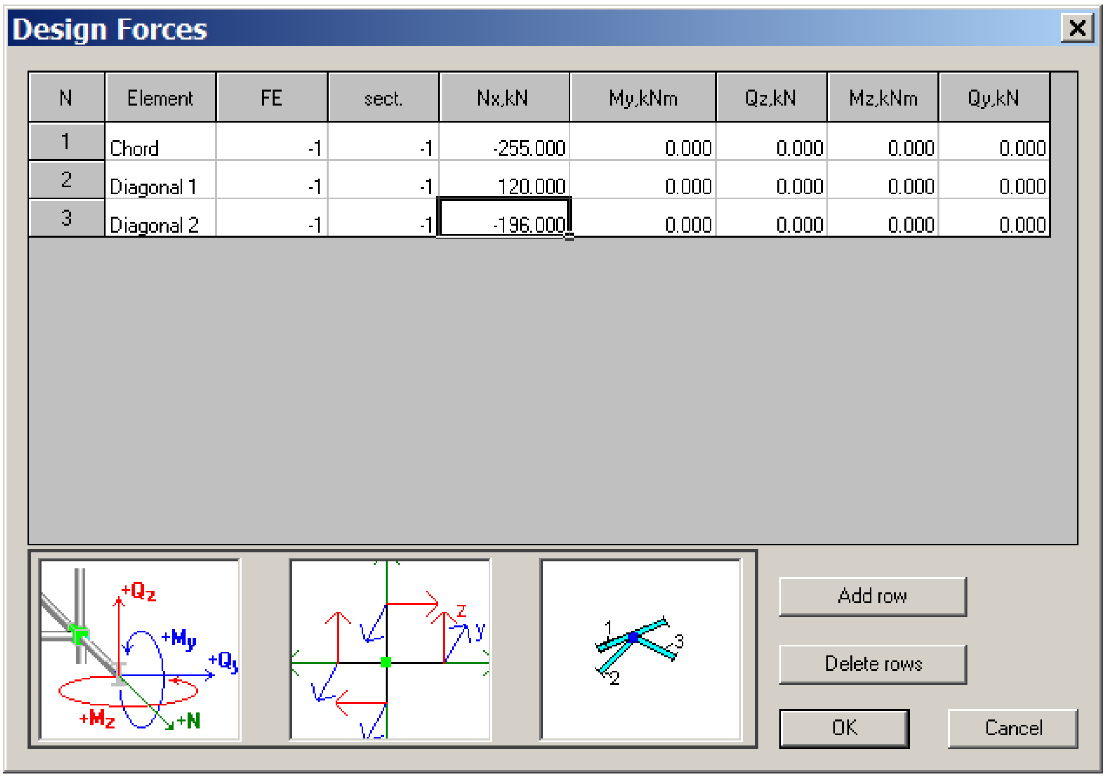
Input of design forces

7. In the Design Forces window define:

• Chord: –255 kN.

• Diagonal 1: 120 kN.

• Diagonal 2: –196 kN.



Design Forces

N	Element	FE	sect.	Nx,kN	My,kNm	Qz,kN	Mz,kNm	Qy,kN
1	Chord	-1	-1	-255.000	0.000	0.000	0.000	0.000
2	Diagonal 1	-1	-1	120.000	0.000	0.000	0.000	0.000
3	Diagonal 2	-1	-1	-196.000	0.000	0.000	0.000	0.000

Figure 13.4 – Input of design forces

Geometry and cross-section definition

8. Open: Edit → Joint Parameters.

9. For the Chord:

   • Profile: seamless hot-rolled tube.

   • Dimension: Ø152 × 6 mm.

10. For Diagonal 1:

   • Ø63.5 × 5 mm.

11. For Diagonal 2:

   • Ø73 × 5 mm.

12. Define lengths:

   • Chord: 3 015 mm.

   • Diagonal 1: 2121 mm.

   • Diagonal 2: 2121 mm.

13. Define inclination angles:

   • Chord: –5.42°.

   • Diagonal 1: –135°.

- Diagonal 2: –44.59°.

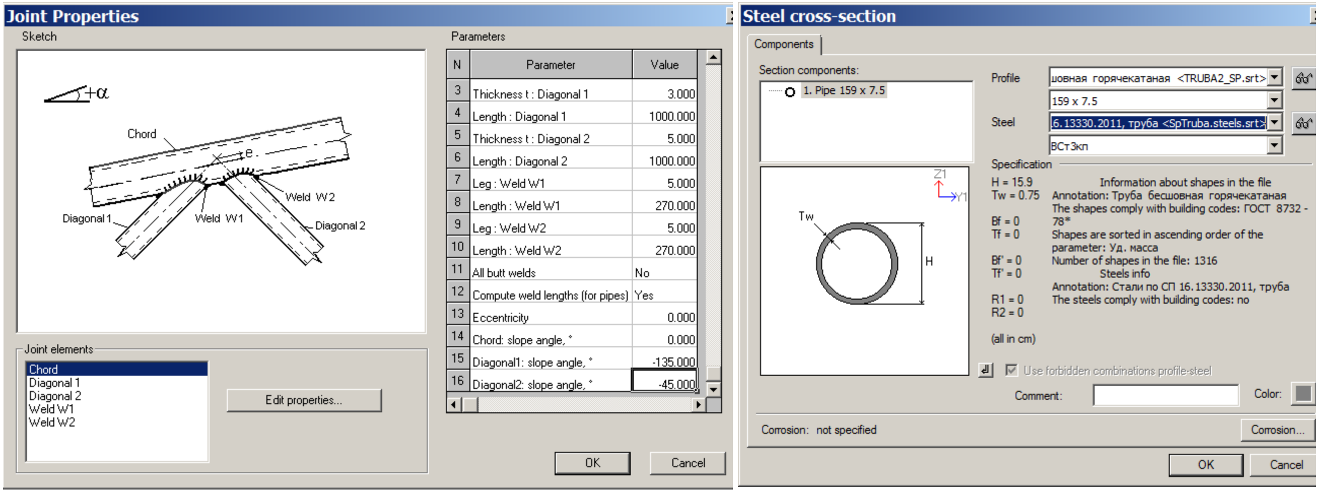


Figure 13.5 – Definition of geometry and cross-sections

Analysis of results and adjustments

14. After verification it was established that the elements do not meet the strength requirements.

Truss joint 1 : Check Results (SP 16.13330.2011)

Parameter	Property	Value	Utilization ratio,%	Internal forces				
				N, kN	My, kNm	Qz, kN	Mz, kNm	Qy, kN
Chord	Thickness t	7.5 mm	111.3	-255.000*	0.000	0.000	0.000	0.000
	Length	3015.0 mm						
Diagonal 1	Thickness t	5.0 mm	100.4	120.000*	0.000	0.000	0.000	0.000
	Length	2121.0 mm						
Diagonal 2	Thickness t	5.0 mm	111.5	-196.000*	0.000	0.000	0.000	0.000
	Length	2121.0 mm						
Weld W1	Leg	5.0 mm	92.2	120.000*	0.000	0.000	0.000	0.000
	Length	231.0 mm						
Weld W2	Leg	5.0 mm	110.4	-196.000*	0.000	0.000	0.000	0.000
	Length	315.0 mm						
All butt welds	--	No	--	--	--	--	--	--
Compute weld lengths (for pipes)	--	Yes	--	--	--	--	--	--
Eccentricity	--	0.0 mm	--	--	--	--	--	--
Chord: slope angle, °	--	-5	--	--	--	--	--	--
Diagonal1: slope angle, °	--	-135	--	--	--	--	--	--
Diagonal2: slope angle, °	--	-44	--	--	--	--	--	--

Figure 13.6 – Verification results before adjustments

15. Perform adjustments:

- Chord: Ø152 × 8 mm.
- Diagonal 1: Ø63.5 × 5.5 mm.
- Diagonal 2: Ø73 × 6 mm.

• Weld throat sizes: № 1 – 5 mm, № 2 – 6 mm.

16. Repeat verification.

Truss joint 1 : Check Results (SP 16.13330.2011)									
Parameter	**Property**	**Value**	**Utilization ratio,%**	**Internal forces**					
				N, kN	**My, kNm**	**Qz, kN**	**Mz, kNm**	**Qy, kN**
Chord	Thickness t	8.0 mm	99.6	-255.000*	0.000	0.000	0.000	0.000
	Length	3015.0 mm						
Diagonal 1	Thickness t	5.5 mm	90.7	120.000*	0.000	0.000	0.000	0.000
	Length	2121.0 mm						
Diagonal 2	Thickness t	6.0 mm	91.4	-196.000*	0.000	0.000	0.000	0.000
	Length	2121.0 mm						
Weld W1	Leg	5.0 mm	92.2	120.000*	0.000	0.000	0.000	0.000
	Length	231.0 mm						
Weld W2	Leg	6.0 mm	92.0	-196.000*	0.000	0.000	0.000	0.000
	Length	315.0 mm						
All butt welds	--	No	--	--	--	--	--	--
Compute weld lengths (for pipes)	--	Yes	--	--	--	--	--	--
Eccentricity	--	0.0 mm	--	--	--	--	--	--
Chord: slope angle, °	--	-5	--	--	--	--	--	--
Diagonal1: slope angle, °	--	-135	--	--	--	--	--	--
Diagonal2: slope angle, °	--	-44	--	--	--	--	--	--

Figure 13.7 – Verification results after adjustments

Self-assessment questions

1. Which key parameters must be specified when creating a new truss joint in LIRA-FEM?

2. How do the coefficients γ_n and γ_c affect the verification results?

3. What is the procedure for assigning risk factors?

4. What are the main reasons for failure to satisfy the strength condition of the joint?

5. How does modifying the geometry or wall thickness of tubes influence the load-bearing capacity?

PRACTICAL CLASS 14

ANALYSIS OF A SPATIAL FRAME ON A PILE FOUNDATION

The aim of this work is to master the methodology of modelling spatial frames on pile foundations using modern computational software. The task involves acquiring skills in applying elastic supports to model piles and their interaction with the soil base, ensuring the correct representation of the spatial behaviour of the "frame–foundation–soil" system.

An important component is the training in performing calculations and analysing displacements, internal forces in the piles and in the frame elements, which allows for a comprehensive assessment of the stress–strain state of the building and the effectiveness of the foundation system.

Initial Data

• Single-span, single-storey building. Span length – 6 m, column spacing – 6 m, storey height – 4 m. Side overhangs of the pile cap slab – 1 m.

• Member cross-sections:

1. Beams – T-section, height 500 mm (flange width – 500 mm, flange thickness – 200 mm, web thickness – 300 mm).
2. Columns – rectangular section, 400 × 400 mm.
3. Roof slab – thickness 200 mm.
4. Pile cap slab – thickness 500 mm.

• Structural elements: columns, beams, floor slabs, foundation slab.
• Material: concrete C32/40, reinforcement A500C, piles – reinforced concrete.

• Loads: permanent, imposed, snow, wind.

• Type of pile foundation: bored or driven piles with specified soil characteristics.

Execution Algorithm

Creating a new task

1. Set the system of units – metres, kN.

2. Create a spatial task.

Geometric model

3. Input node coordinates according to the plan and storey height.

4. Define bar elements (columns, beams) and plate elements (floor slabs, foundation slab).

5. Ensure the correctness of the spatial scheme (no isolated nodes).

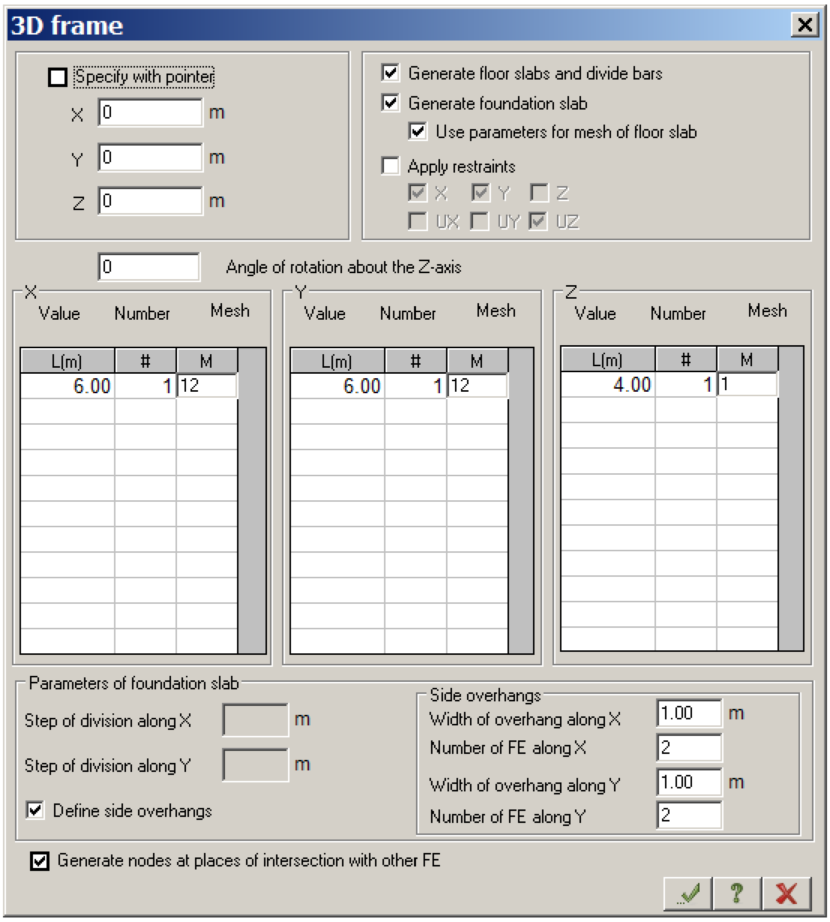


Figure 14.1 – Creation of the geometric model

Assignment of materials and cross-sections

6. Define the physical and mechanical characteristics of concrete and reinforcement.

7. For the foundation slab and piles, assign the relevant cross-sections.

8. If necessary, define physically non-linear material properties.

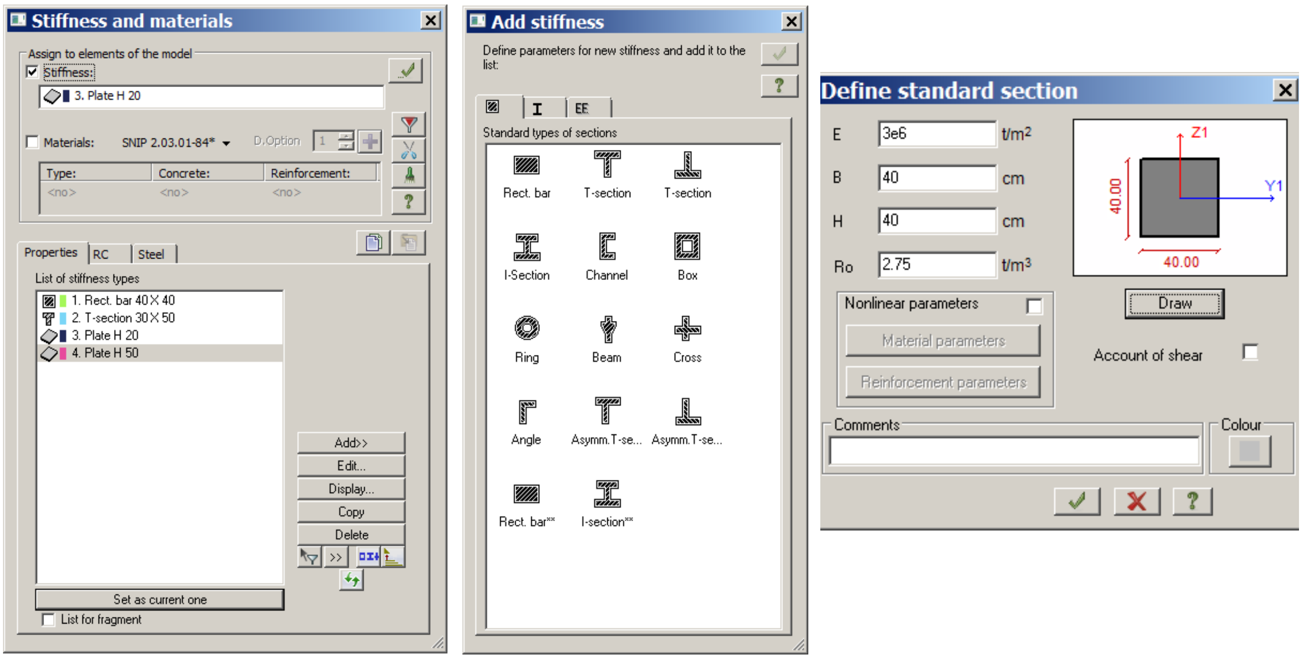


Figure 14.2 – Assignment of stiffness and cross-sections

Modelling of pile foundation

9. For each pile, introduce an elastic support element with stiffness parameters (subgrade coefficients determined by calculation or according to standards).

10. Arrange the piles according to the foundation plan.

11. If necessary, model piles as bar elements with fixed nodes in the soil.

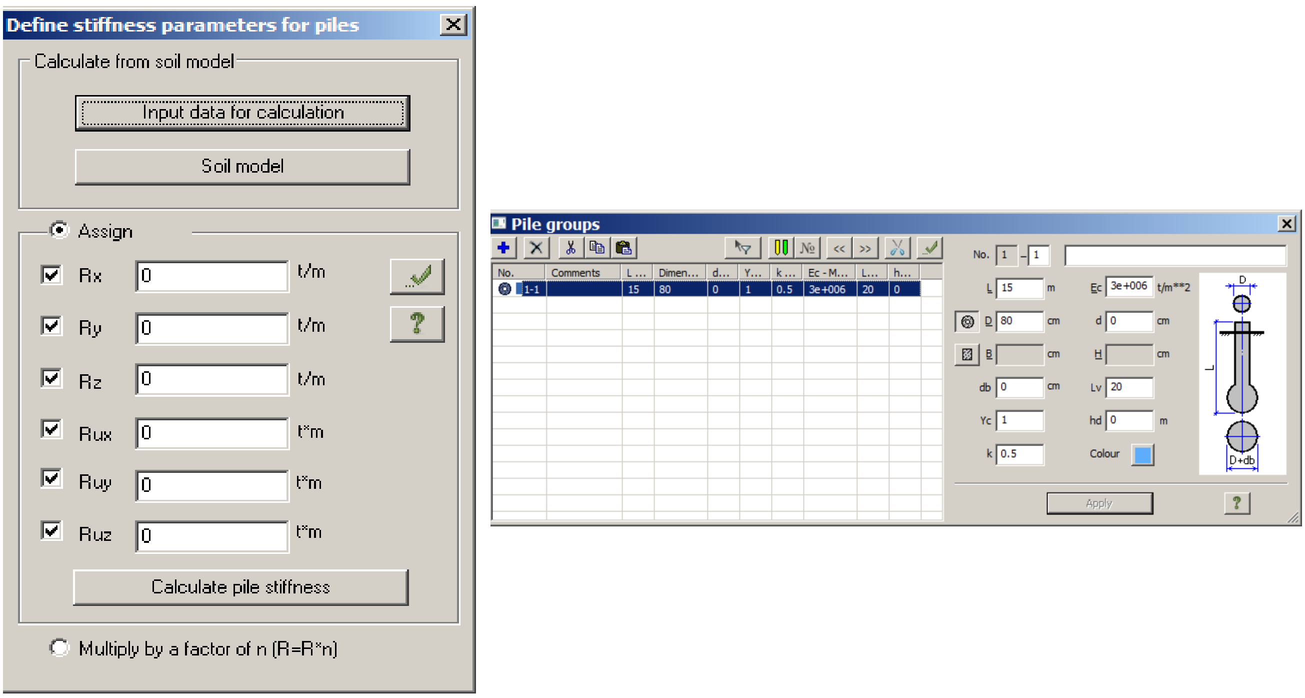


Figure 14.3 – Assignment of pile stiffness parameters

Boundary conditions

12. Restrain the bottom nodes of the piles (full fixity or elastic conditions, depending on the model).

13. At the top nodes of the piles, assign rigid connections with the foundation slab.

Load assignment

14. Permanent: self-weight of the structure, weight of infill.

15. Imposed: service loads on floors.

16. Climatic: snow, wind.

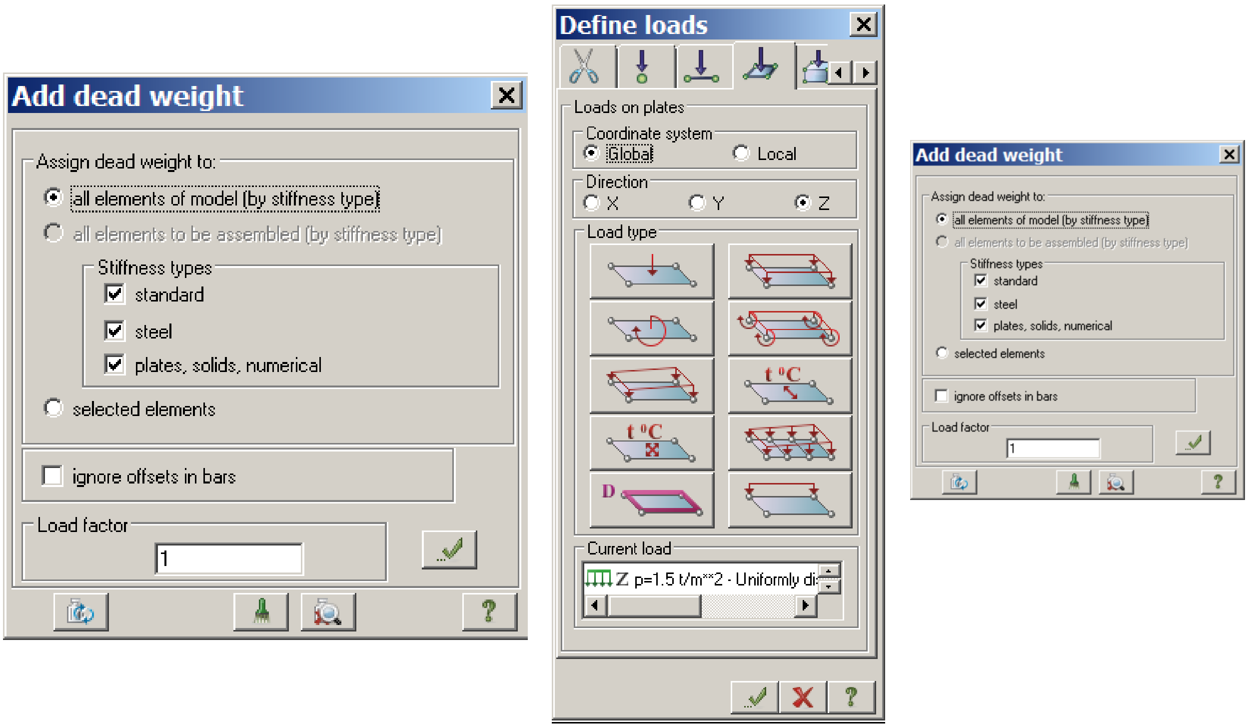


Figure 14.4 – Load assignment

Load combinations

17. ULS – for verification of load-bearing capacity.

18. SLS – for assessment of serviceability.

Execution of calculation

19. Run the model analysis.

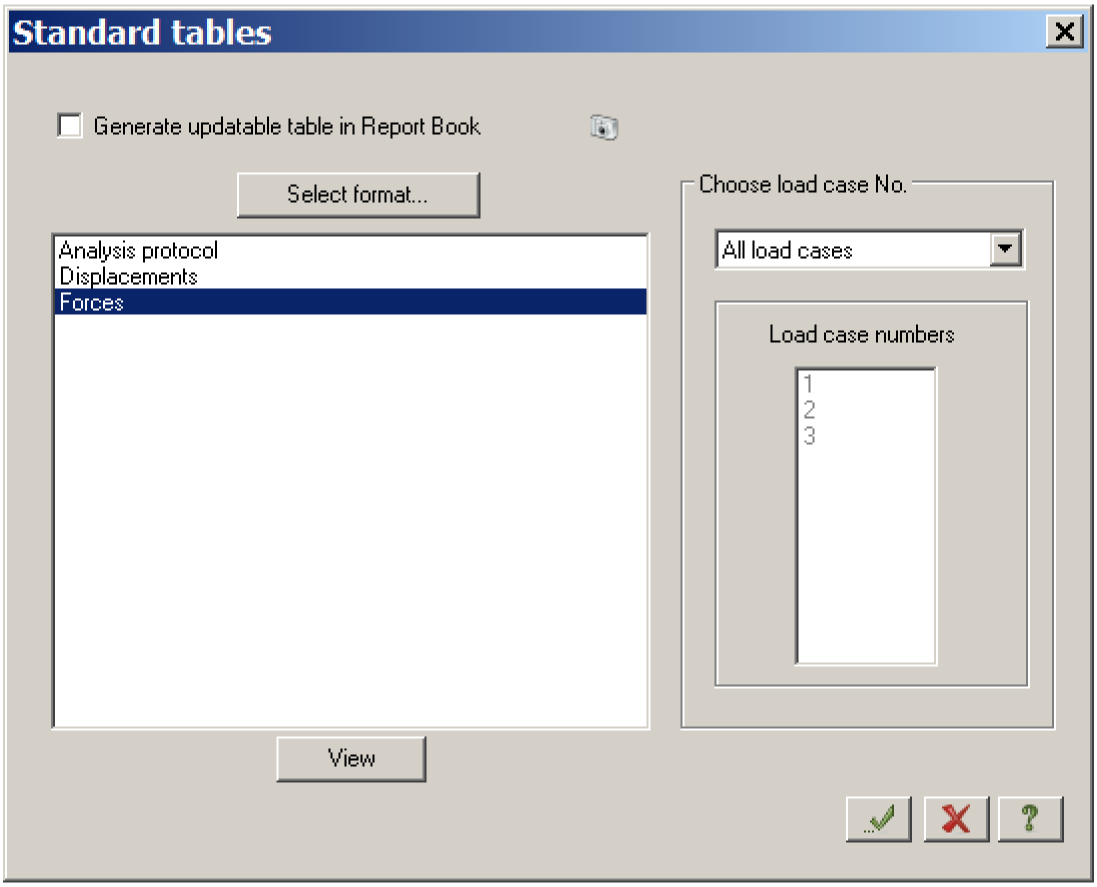


Figure 14.5 – Formation and review of calculation result tables

20. Obtain nodal displacements and forces in piles and structural members.

21. Determine maximum pile forces and compare them with the design bearing capacity.

Result analysis

22. Verify displacements and deflections against the allowable limits.

23. Analyse the uniformity of load distribution among piles.

24. Determine whether modifications in pile arrangement or quantity are required.

Self-assessment questions

1. How are piles assigned as elastic supports in the LIRA-FEM model?

2. What is the difference between modelling a pile as a bar element and as an elastic support?

3. Which parameters define the stiffness of a pile in the calculation?

4. How can the uniformity of load distribution among piles be checked?

5. Which load combinations are used to verify load-bearing capacity and serviceability?

PRACTICAL CLASS 15

MODELLING OF A COMPOSITE (STEEL-CONCRETE) CROSS-SECTION

The aim of this work is to master the principles of creating and analysing composite steel-concrete cross-sections in the LIRA-FEM software package. The task involves learning how to define the joint action of steel and concrete within one section to correctly model their interaction under service loads.

An additional objective is to acquire skills in verifying the load-bearing capacity and stiffness of composite elements in accordance with current design standards, thus ensuring the reliability and efficiency of structural performance.

Initial Data

• Cross-section of a composite slab with a monolithic concrete slab cast on a profiled steel deck (figure 15.1):

1. Reinforced concrete slab: material – concrete C25/30; reinforcement: DSTU 9130:2021, class A400C, Ø20 mm.

2. Profiled steel sheet with trapezoidal corrugation: height h = 116 mm, B1 = 187 mm, t = 1.2 mm; material – galvanised steel.

3. Beam: asymmetric I-section (upper flange – 150 × 12 mm; lower flange – 630 × 10 mm; web – 320 × 16 mm); material – rolled steel.

• Materials: concrete class C32/40, steel class S355.

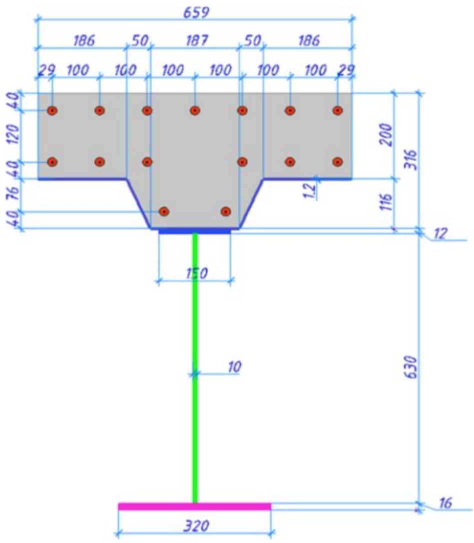


Figure 15.1 – Cross-section of a composite floor slab

Execution Algorithm

Creating a new task and section

1. Launch the «Sections» module in LIRA-FEM.

2. Select the type of section – composite (steel-concrete).

3. Input the main dimensions of the concrete part (height, width, flange thickness, etc.).

Adding steel elements

4. Select the type of steel profile (I-section, channel, plate).

5. Specify its position within the concrete part – coordinates, orientation.

6. Define the steel grade.

Assignment of materials

7. For concrete – assign modulus of elasticity, compressive strength, density.

8. For steel – assign modulus of elasticity, yield strength, density.

9. Verify the system of units.

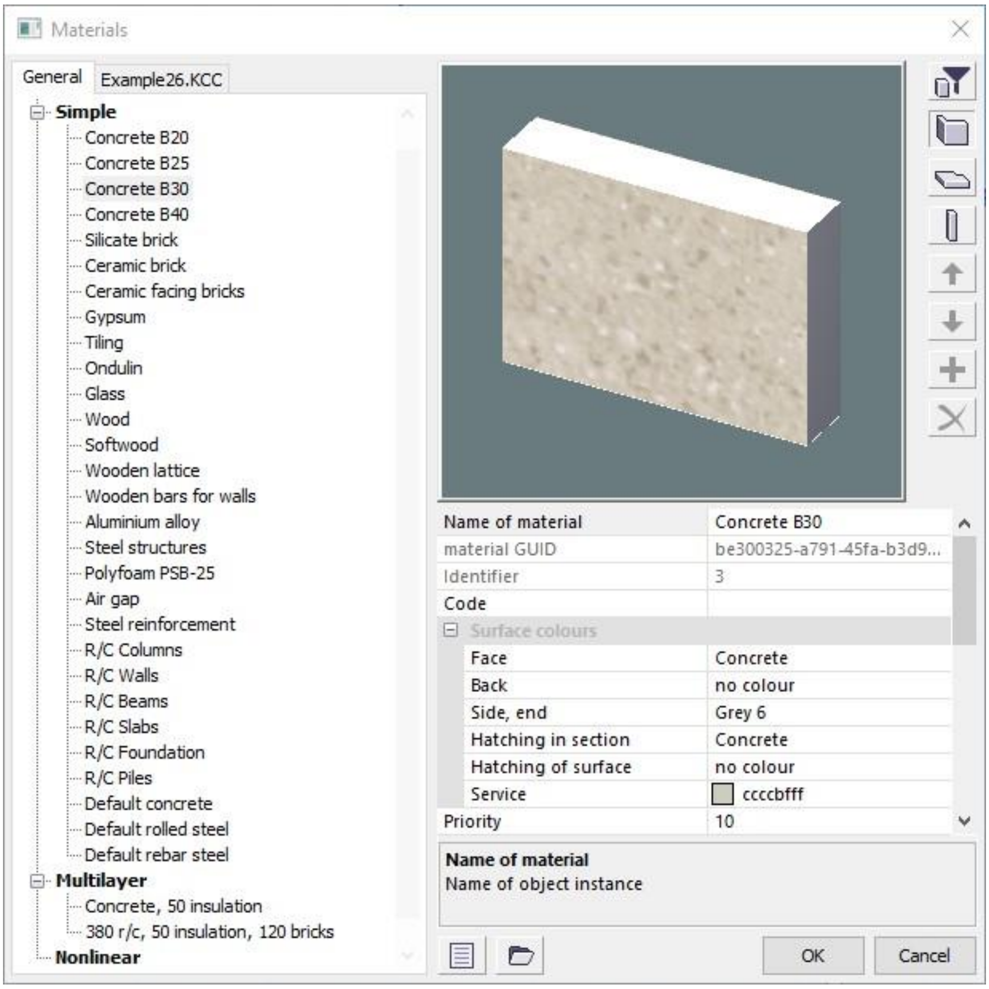


Figure 15.2 – Material assignment

Assignment of reinforcement

10. If the section contains additional reinforcement – input the bar diameter, quantity, and arrangement.

11. Assign the reinforcement grade.

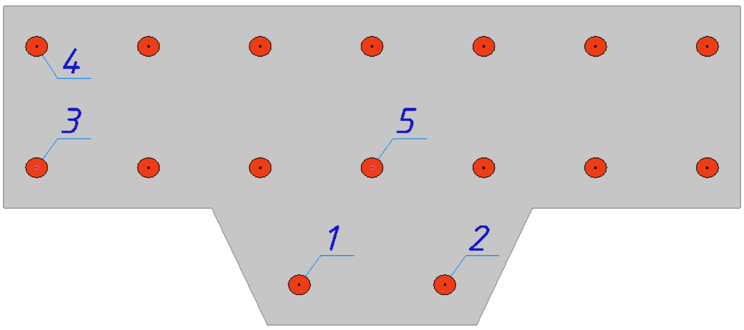


Figure 15.3 – Section outline with reinforcement

Assignment of design forces

12. Input values of N, M_x, M_y, Q_x, Q_y according to the results of static analysis.

13. Ensure that the signs of forces correspond to the adopted convention in the program (compression – “+”, tension – “–”).

Verification procedure

14. Run the section analysis.

15. Review results: utilisation factors, stresses in concrete and steel, deflections.

Cross-section properties

Copy | Copy all

Annotation	Value	Units	Name
Geometric properties of whole cross-section			
Xo	0	mm	X-coordinate of gravity centre in the current coordinate system
Yo	0	mm	Y-coordinate of gravity centre in the current coordinate system
ϕ	0.01	°	Rotation of principal Y1-axis of the section relative to x-axis of current coordinate system
Ry	314.07	mm	Gyration radius relative to principal Y1-axis
Rz	152.81	mm	Gyration radius relative to principal Z1-axis
Pext	3809.45	mm	Perimeter of outer contours
Pint	0	mm	Perimeter of inner contours
Ro	3.064	t/m³	Average density of cross section
g	0.546	tf/m	Average unit weight
Y-	70.72	mm	Core distance in negative direction of principal Y1-axis
Y+	70.78	mm	Core distance in positive direction of principal Y1-axis
Z-	318.84	mm	Core distance in negative direction of principal Z1-axis
Z+	148.39	mm	Core distance in positive direction of principal Z1-axis
Torsional properties			
Yt	17.13	mm	Y1 coordinate of torsion centre in coordinate system of principal axes Y1oZ1
Zt	110.99	mm	Z1 coordinate of torsion centre in coordinate system of principal axes Y1oZ1
Shear properties			
Ys	16.62	mm	Y1-coordinate of shear centre in coordinate system of principal axes Y1oZ1
Zs	118.15	mm	Z1-coordinate of shear centre in coordinate system of principal axes Y1oZ1
Stiffness properties			
EA	9.1786e5	tf	Axial stiffness
EIu	90539	tf·m²	Bending stiffness relative to central U-axis
EIv	21432	tf·m²	Bending stiffness relative to central V-axis
EIuv	-17.23	tf·m²	Centrifugal stiffness relative to central axes UV
EIy	90539	tf·m²	Bending stiffness relative to principal Y1-axis
EIz	21432	tf·m²	Bending stiffness relative to principal Z1-axis
ESy	1.1431e5	tf·m	Product of static moment of half-section and its elasticity modulus relative to principal Y1-axis
ESz	54274	tf·m	Product of static moment of half-section and its elasticity modulus relative to principal Z1-axis
GJt	4248.9	tf·m²	Torsional stiffness - product of shear modulus and torsion moment of inertia
EIw	676.157354	tf·m⁴	Warping stiffness - product of elasticity modulus and warping moment
GFy	2.3998e5	tf	Shear stiffness Y1 - product of shear modulus and shear area relative to principal Y1-axis
GFz	21461	tf	Shear stiffness Z1 - product of shear modulus and shear area relative to principal Z1-axis

Figure 15.4 – Review and analysis of section characteristics

Analysis and conclusions

16. If utilisation factors > 1 – modify dimensions or materials.

17. Analyse the compatibility of deformations between steel and concrete.

18.Draw conclusions regarding the efficiency of the section.

Self-assessment questions

1. What are the main differences in behaviour between composite steel-concrete elements and conventional reinforced concrete elements?

2. How is the joint action of steel and concrete defined in LIRA-FEM?

3. Which parameters govern the load-bearing capacity of a composite section?

4. Why is it important to check deformation compatibility?

5. How does the positioning of steel elements in concrete influence the stiffness and strength of the section?

PRACTICAL CLASS 16

FRAME STABILITY UNDER PROGRESSIVE COLLAPSE

The aim of this work is to master the methodology of modelling progressive collapse in spatial frames using the LIRA-FEM software package. The task involves learning how to apply special calculation scenarios to assess the loss of stability of a structure following accidental damage to individual elements.

An important component is understanding the impact of element removal on force redistribution, changes in the stress-strain state, and the overall load-bearing capacity of the system. This enables an assessment of the reliability and safety of the building under accidental loading scenarios.

Initial Data

- Materials: steel (S355), concrete (C32/40).
- Loads (characteristic values):
    1. Floor slabs: 6 m × 0.33 t/m² = 1.98 t/m.
    2. Partitions: 6 m × 0.092 t/m² = 0.552 t/m.
    3. Floor construction: 6 m × 0.072 t/m² = 0.432 t/m.
    4. Imposed load: 6 m × 0.15 t/m² = 0.9 t/m.
    5. Wind: 0.023 t/m².
- Loads (design values):
    6. Floor slabs: 1.98 t/m × 1.1 = 2.18 t/m.

7. Partitions: 0.552 t/m × 1.2 = 0.662 t/m.

8. Floor construction: 0.432 t/m × 1.3 = 0.562 t/m.

9. Imposed load: 0.9 t/m × 1.3 = 1.17 t/m.

10. Wind: 0.023 t/m² × 1.4 = 0.032 t/m².

• Damage conditions: removal of one or several elements (columns, beams) in the model.

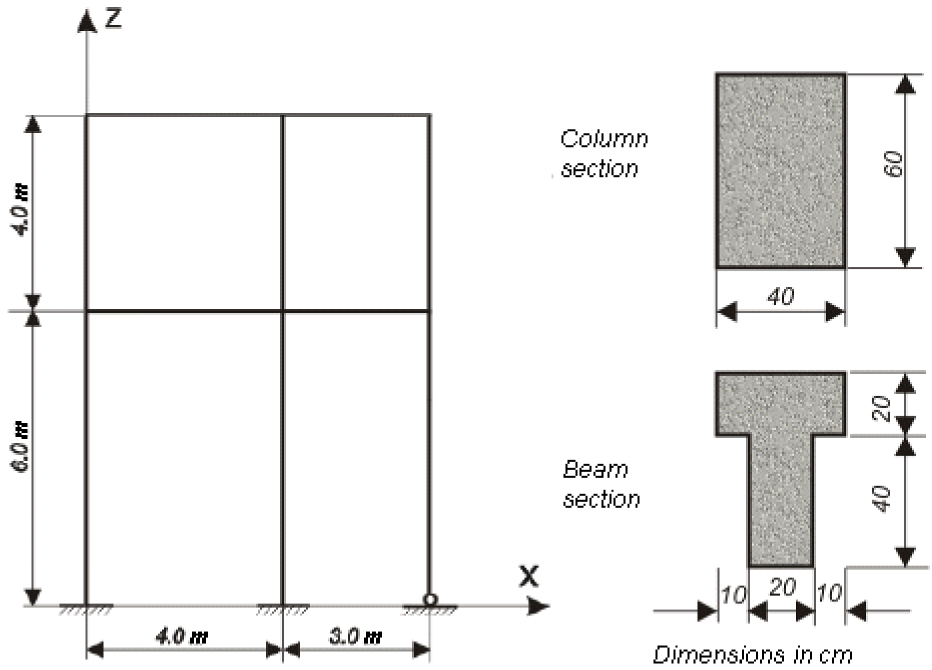


Figure 16.1 – Initial data

Execution Algorithm

Creation and preparation of the base model

1. Launch LIRA-FEM and create a new spatial frame model.

2. Define the building geometry.

3. Assign rigid diaphragms for floors (if required) to ensure correct load distribution.

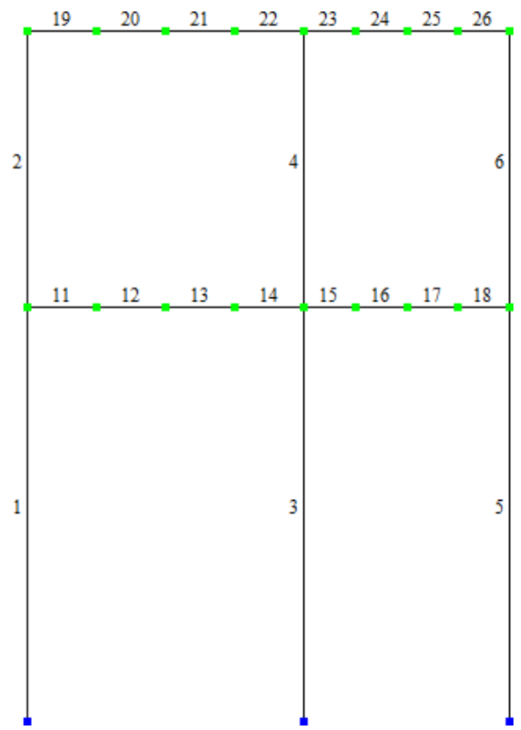


Figure 16.2 – General view of the frame after element addition

Assignment of element properties

4. Assign cross-sections and materials (steel / concrete) to columns, beams, and slabs.

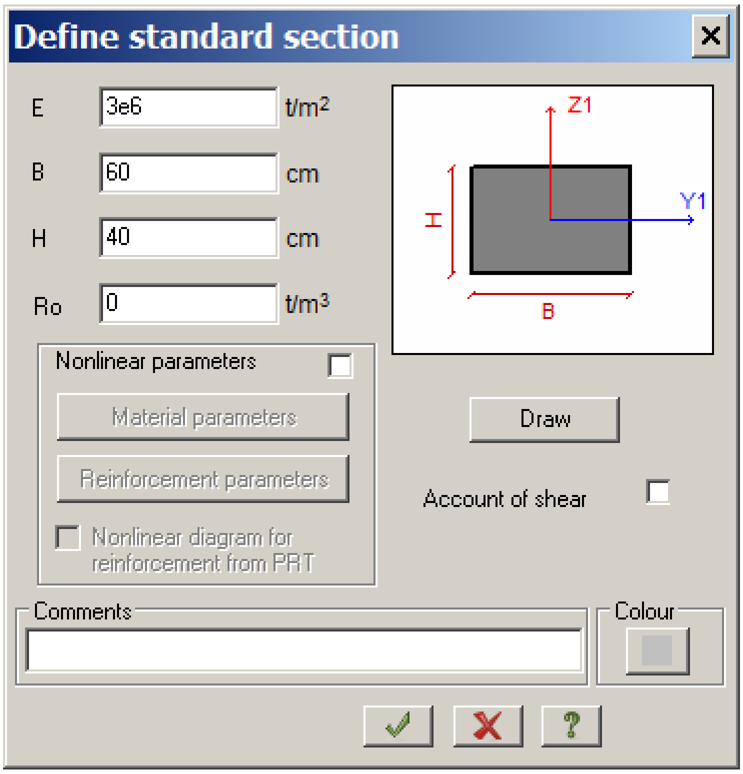


Figure 16.3 – Standard section assignment

5. For each material, specify modulus of elasticity, density, and design strength.

6. Assign boundary conditions for column bases according to the structural scheme.

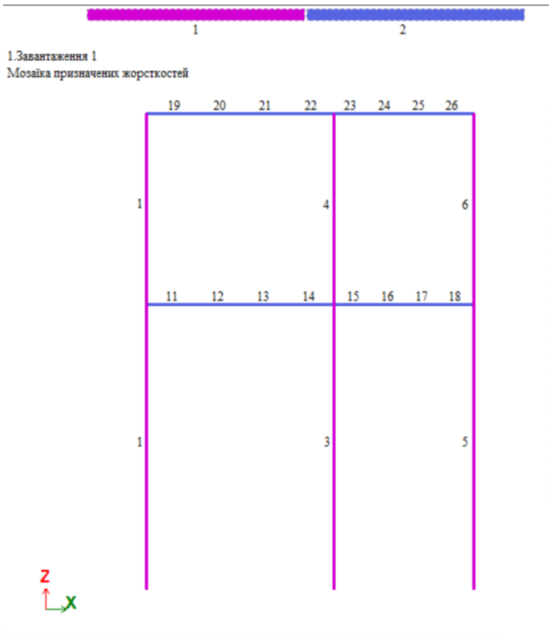


Figure 16.4 – Mosaic of assigned stiffnesses

Load assignment

7. Automatically include the self-weight of structures.

8. Add permanent and variable loads (imposed, snow, wind) according to design standards.

9. Generate the required load combinations (LC) in compliance with codes.

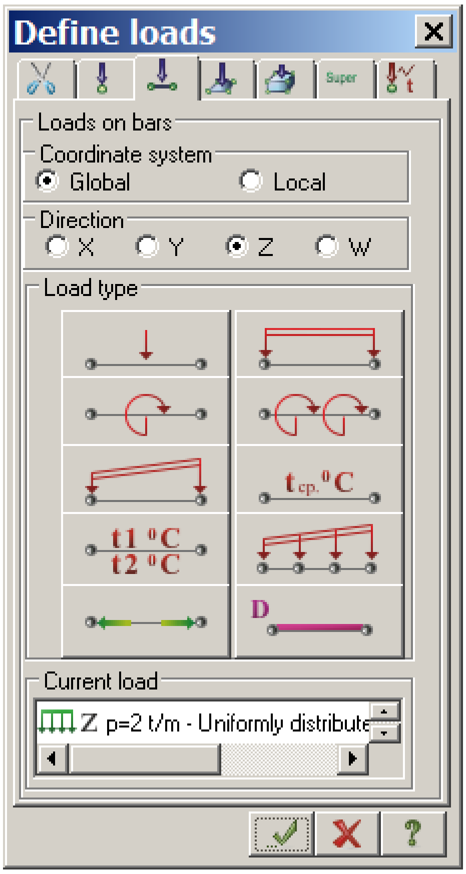


Figure 16.5 – “Load assignment” dialogue window

Base model calculation

10. Run the analysis in a linear–elastic setting.

11. Verify displacements, internal forces, and stresses in elements.

12. Save the results as a reference state for comparison.

Accidental damage simulation

13. Identify the element(s) to be “removed” for simulating an accidental scenario (e.g., middle column at ground floor).

14. Delete the selected element or replace its material with “very soft” (near-zero stiffness) to simulate failure.

15. Ensure the geometry of the model remains consistent.

Recalculation

16. Run the analysis of the model with the removed element.

17. Compare displacements, forces, and stresses with the baseline condition.

18. Check whether allowable deflections and stresses are exceeded.

Redistribution of forces analysis

19. Assess how adjacent elements (columns, beams, slabs) respond after removal of one element.

20. Identify which elements become “critical” after damage.

21. Consider whether additional strengthening of the structure is required.

Conclusions

22. Formulate a conclusion regarding the frame’s stability against progressive collapse.

23. If the system is unstable – suggest measures to improve robustness (redundancy of load-bearing elements, additional bracing, etc.).

Self-assessment questions

1. What is progressive collapse and what are its main causes?

2. How can accidental element failure be modelled in LIRA-FEM?

3. Why is it important to analyse force redistribution after damage?

4. What measures can enhance structural resistance to progressive collapse?

5. How should the results of comparing the baseline and damaged states be interpreted?

REFERENCE LIST

1. Програмний комплекс ЛІРА-САПР. Приклади розрахунку і проєктування [Електрон. ресурс] : навч. посіб. – Електрон. текст. дані. – Київ : Ліра-Ленд, 2022. – 635 с. – Режим доступу: https://www.liraland.ua/download/private /lira/2023/lira_sapr_examples_ua.pdf, вільний (дата звернення: 22.08.2025). – Назва з екрана.

2. ДБН В.1.2-2:2006. Навантаження і впливи. Норми проєктування : зі змінами № 1 та № 2 [Електрон. ресурс]. – Чинний від 2007–01–01. – Електрон. текст. дані. – Київ : Міністерство регіонального розвитку та будівництва України, 2020. – 72 с. – Режим доступу: https://e-construction.gov.ua/files/new_doc/3070 574263435003317/2023-03-31/670e67af-b4c0-4a2f-b855a6a41520d31f.pdf, вільний (дата звернення: 22.08.2025). – Назва з екрана.

3. ДБН В.2.6-98:2009. Бетонні та залізобетонні конструкції. Основні положення : із зміною № 1 [Електрон. ресурс]. – Чинний від 2020–06–01. – Електрон. текст. дані. – Київ : Міністерство розвитку громад та територій України, 2020. – 71 с. – Режим доступу: https://econstruction.gov.ua/files/new_doc/3080063 210573792873/2023-04-13/adb4ca1e-8595-4d35-9b22-a858d85864b4.pdf, вільний (дата звернення: 22.08.2025). – Назва з екрана.

4. ДСТУ Б В.2.6-156:2010. Бетонні та залізобетонні конструкції з важкого бетону. Правила проєктування [Електрон. ресурс]. – Чинний від 2011–06–01. – Електрон. текст. дані. – Київ : Мінрегіонбуд України, 2011. – 123 с. – Режим доступу: https://dbn.co.ua/load/normativy/dstu/5-1-0-800, вільний (дата звернення: 22.08.2025). – Назва з екрана.

5. ДБН В.2.6-198:2014. Сталеві конструкції. Норми проєктування : зі зміною № 1 [Електрон. ресурс] – Чинний від 2022–09–01. – Електрон. текст. дані. – Київ : Міністерство розвитку громад та територій України, 2022. – 105 с. – Режим доступу: https://e-construction.gov.ua/files/new_doc/3022132243112920267/2023-01-24/7ba57e35-6fb0-40fc-bf78-dfd412593a0f.pdf, вільний (дата звернення: 22.08.2025). – Назва з екрана.

6. ДБН В.2.6-160:2010. Сталезалізобетонні конструкції. Основні положення : зі зміною № 1 [Електрон. ресурс]. – Чинний від 2020–06–01.– Київ : Міністерство розвитку громад та територій України, 2020. – 54 с. – Режим доступу: https://e-construction.gov.ua/files/new_doc/3200413563881522897/2023-09-26/9641a3ce-3e67-4a75-aa19-717d7f6f0764.pdf, вільний (дата звернення: 22.08.2025). – Назва з екрана.

Електронне навчальне видання

Методичні рекомендації

до проведення практичних занять та організації самостійної роботи з навчальної дисципліни

«КОМП'ЮТЕРНІ МЕТОДИ РОЗРАХУНКУ БУДІВЕЛЬНИХ КОНСТРУКЦІЙ»

(для здобувачів другого (магістерського) рівня вищої освіти всіх форм навчання зі спеціальності 192 – Будівництво та цивільна інженерія, освітня програма «Промислове та цивільне будівництво»)

(Англ. мовою)

Укладачі: **РЕЗНІК** Петро Аркадійович,
ЛУГЧЕНКО Олена Іванівна

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

План 2025, поз. 582М

Підп. до друку 09.02.2026. Формат 60 × 84/16.
Ум. друк. арк. 5,4.

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.