



Co-funded by
the European Union

The **Bridge**
IM architecture
engineering
construction

Case Study

MODERN APPROACHES to Design, Analysis and Assessment of Building Systems Using BIM-Technologies



<https://thebridgeproject.org.ua/>

**MODERN APPROACHES TO THE DESIGN, ANALYSIS AND EVALUATION OF
BUILDING SYSTEMS USING BIM TECHNOLOGIES**

Case studies

Edited by Yu.V. Bulhakova, N. S. Chernova

Dnipro

2025

UDC 624.01:004.942(075.8)

M 78

The case studies have been recommended for publication by the Academic Council of the Ukrainian State University of Science and Technologies, Ministry of Education and Science of Ukraine, Minutes No. 1 dated September 24, 2025.

Team of authors: Bolotov M. H., Hanieiev T. R., Davydov I. I., Zahilskyi V. A., Kovba V. V., Levkivskyi D. V., Malakhov V. V., Moskvitina A. S., Pozniak O.R., Pushchina N. V., Rybachov S. H., Rudenko M. M., Siedin V. L., Shekhovtsov V. I., Soroka M. M., Surianinov M. H., Tytarenko R. Yu., Khmil R.Ye., Chaban V. P.

Edited by Yu.V. Bulhakova, N. S. Chernova

Academic editors:

Yu.V. Bulhakova, Candidate of Technical Sciences, Associate Professor, Department of Railway Commercial Operations Management, National Transport University (Kyiv).

N.S. Chernova, Candidate of Economic Sciences, Associate Professor, Department of Economics and Management, Ukrainian State University of Science and Technologies (Dnipro).

Reviewers:

I. I. Nazarenko, President of the Academy of Construction of Ukraine, Doctor of Technical Sciences, Professor, Department of Machines and Equipment of Technological Processes, Kyiv National University of Construction and Architecture, Honoured Worker of Science and Technology of Ukraine (Kyiv).

M. M. Babenko, Candidate of Technical Sciences, Associate Professor, Department of Materials Engineering and Physic, Slovak University of Technology in Bratislava.

Modern approaches to the design, analysis and evaluation of building systems using BIM technologies : Practical manual / M. H. Bolotov, T. R. Hanieiev, I. I. Davydov et al.; edited by Yu. V. Bulhakova and N. S. Chernova ; Ukrainian State University of Science and Technologies. – Electronic edition. – Dnipro : USUST, 2025. – 240 p.

ISBN 978-617-8314-81-1 (PDF)

The case studies present materials from scientific and practical studies carried out within the framework of the international project of the European Union's Erasmus+ programme 'The BRIDGE – Bridging Innovations, Knowledge and Skills for Integrating Digitalization in the Ukrainian Construction Sector'.

It focuses on the use of BIM technologies, as well as on innovative solutions aimed at ensuring energy efficiency, analysis accuracy, and interoperability of software environments. The materials cover the issues of thermal efficiency in eco-oriented modular buildings, modelling of historical structures based on point cloud data, the application of BIM in structural design, the analysis of steel and reinforced concrete structures, the analysis of soil-foundation interaction, and the development of indoor microclimate models for public buildings.

The publication is intended for academic and research staff, postgraduate students, university students, as well as experts in the fields of construction, architecture, energy, and digital modelling who are interested in the practical application of BIM technologies in education and professional practice.

UDC 624.01:004.942(075.8)



This work is licensed under Creative Commons License

[«Attribution-NonCommercial-ShareAlike» 4.0 International \(CC BY-NC-SA 4.0\)](https://creativecommons.org/licenses/by-nc-sa/4.0/)

ISBN 978-617-8314-81-1 (PDF)

© Bolotov M. H., Hanieiev T. R et al., 2025

© Ukrainian State University of Science and Technologies, 2025

TEAM OF AUTHORS

M. H. Bolotov, Candidate of Technical Sciences, Associate Professor, Department of Welding and Construction Technologies, National University 'Chernihiv Polytechnic'

T. R. Hanieiev, Candidate of Technical Sciences, Associate Professor, Department of Welding and Construction Technologies, National University 'Chernihiv Polytechnic'

I. I. Davydov, Candidate of Technical Sciences, Associate Professor, Department of Metal, Timber and Plastic Structures, Ukrainian State University of Science and Technologies, SEI 'Prydniprovsk State Academy of Civil Engineering and Architecture'

V. A. Zahilskyi, Candidate of Technical Sciences, Associate Professor, Department of Engineering Geology and Geotechnics, Ukrainian State University of Science and Technologies, SEI 'Prydniprovsk State Academy of Civil Engineering and Architecture'

V.V. Kovba, Candidate of Technical Sciences, Associate Professor, Department of Engineering Geology and Geotechnics, Ukrainian State University of Science and Technologies, SEI 'Prydniprovsk State Academy of Civil Engineering and Architecture'

D. V. Levkivskyi, Candidate of Technical Sciences, Associate Professor, Department of Strength of Materials, Kyiv National University of Construction and Architecture

V. V. Malakhov, Candidate of Technical Sciences, Associate Professor, Department of Reinforced Concrete Structures and Transport Facilities, Odesa State Academy of Civil Engineering and Architecture

A. S. Moskvitina, Candidate of Technical Sciences, Associate Professor, Department of Heat and Gas Supply and Ventilation, Kyiv National University of Construction and Architecture

O. R. Pozniak, Candidate of Technical Sciences, Associate Professor, Department of Construction Production, Institute of Building and Engineering Systems, Lviv Polytechnic National University

N. V. Pushchina, Assistant Lecturer, Department of Information Technologies and Applied Mathematics, Odesa State Academy of Civil Engineering and Architecture

S. H. Rybachov, Candidate of Technical Sciences, Associate Professor, Professor of the Department of Heat and Gas Supply and Ventilation, Kyiv National University of Construction and Architecture

M. M. Rudenko, Senior Lecturer, Department of Welding and Construction Technologies, National University 'Chernihiv Polytechnic'

V. I. Shekhovtsov, Candidate of Technical Sciences, Associate Professor, Department of Reinforced Concrete Structures and Transport Facilities, Odesa State Academy of Civil Engineering and Architecture

V. L. Siedin, Doctor of Technical Sciences, Professor, Department of Engineering Geology and Geotechnics, Ukrainian State University of Science and Technologies, SEI 'Prydniprovsk State Academy of Civil Engineering and Architecture'

M. M. Soroka, Candidate of Technical Sciences, Associate Professor, Department of Structural Mechanics, Odesa State Academy of Civil Engineering and Architecture

M. H. Surianinov, Doctor of Technical Sciences, Professor, head of the Department of Structural Mechanics, Odesa State Academy of Civil Engineering and Architecture

R. Yu. Tytarenko, Candidate of Technical Sciences, Associate Professor, Department of Building Structures and Bridges, Institute of Civil Engineering, Infrastructure and Life Safety, Lviv Polytechnic National University

R. Ye. Khmil, Doctor of Technical Sciences, Associate Professor, head of the Department of Building Structures and Bridges, Institute of Civil Engineering, Infrastructure and Life Safety, Lviv Polytechnic National University

V. P. Chaban, Candidate of Technical Sciences, Associate Professor, Department of Metal, Timber and Plastic Structures, Ukrainian State University of Science and Technologies, SEI 'Prydniprovsk State Academy of Civil Engineering and Architecture'

CONTENTS

1. MODELING OF HISTORICAL BUILDINGS USING BIM TECHNOLOGIES BASED ON POINT CLOUDS

Hanieiev T. R.

2. THERMAL EFFICIENCY ASSESMENT OF ECO-ORIENTED PREFABRICATED MODULAR BUILDINGS

Bolotov M. H., Rudenko M. M.

3. OPEN BIM-BASED CO-SIMULATION FOR BUILDING ENERGY AND LCA: TWO BUILDINGS IN THE UK AND LUXEMBOURG

Tytarenko R. Yu., Chmil R. Ye., Pozniak O. R.

4. DEVELOPMENT OF 3D MODELS OF MICROCLIMATE FORMATION SYSTEMS OF PUBLIC BUILDINGS USING LINEAR SOFTWARE

Rybachov S. H., Moskvitina A. S.

5. COMPARATIVE CALCULATION OF REINFORCED CONCRETE BEAMS ACCORDING TO UKRAINIAN (STATE BUILDING STANDARDS) AND EUROPEAN (EN 1992-1-1) STANDARDS

Shekhovtsov V. I., Malakhov V. V.

6. CALCULATION OF THE LOAD-BEARING CAPACITY OF FRAMES USING THE LIRA-SAPR SOFTWARE

Surianinov M. H., Soroka M. M., Pushchina N. V.

7. ANALYSIS AND DESIGN OF STEEL STRUCTURES IN TEKLA STRUCTURES SOFTWARE

Davydov I. I., Chaban V.P.

8. BASICS OF CALCULATION AND DESIGN OF BUILDINGS USING LIRA-FEM SOFTWARE

Levkivskyi D. V.

9. MODELING OF SOIL-FOUNDATION INTERACTION WITH PLAXIS 3D SOFTWARE

Kovba V.V., Zahilskyi V. A., Siedin V. L.

10. APPLICATION OF BIM IN STRUCTURAL DESIGN: CASE STUDY ON INTEROPERABILITY ANALYSIS, REINFORCEMENT DETAILING, AND QUANTITY TAKE-OFF

Tytarenko R. Yu., Chmil R. Ye., Pozniak O. R.

FOREWORD

The current stage of Ukraine's development is characterized by complex challenges related to the post-war reconstruction of infrastructure, modernization of the economy, and the transition towards sustainable development. Under these circumstances, innovative educational initiatives are of particular significance, as they unite science, education, technology, and business, shaping a new generation of professionals who think strategically, act sustainably, and possess advanced digital skills.

One of such initiatives is the Erasmus+ CBHE project 'The BRIDGE – Bridging University and Industry through a Master Programme Supporting the Development of Green Jobs and Digital Skills in the Ukrainian Construction Sector' (Project No. 101127884 – ERASMUS-EDU-2023-CBHE). The project is aimed at developing a new generation of Master's programmes, fostering green competences and digital technologies in Ukraine's construction sector.

The development of the construction industry and the training of a new generation of engineers are of key importance for the post-war recovery of the country. The success of infrastructure modernization and the transition to an energy-efficient economy largely depend on the level of professional training, innovation capacity, and the ability to integrate European practices. For this reason, the BRIDGE project has become an important platform that unites universities, enterprises, research institutions, and public authorities around a common goal – building a resilient, modern, and digitally oriented educational ecosystem.

The case studies have been developed within the framework of this project. It reflects the ideas, principles, and practical approaches underlying the modernization of higher education in Ukraine in the context of its integration into the European Higher Education Area.

The project is designed to support Ukraine's post-war recovery, promote the development of green technologies and digital competences, and strengthen cooperation between universities and industry in the construction sector.

The BRIDGE project is implemented by a consortium of universities and organizations from six countries - Ukraine, Poland, Italy, Germany, and Slovakia with the participation of the Ministry of Education and Science of Ukraine. Its goal is to create a modern educational environment that will ensure the training of a new generation of professionals who could implement the principles of sustainable development, energy efficiency, and digitalization in the construction sector.

The need to modernize Ukraine's construction sector has become particularly urgent in the context of post-war reconstruction. More than 60% of buildings in Ukraine require renovation, and their level of energy efficiency remains low. At the same time, the state has declared its commitment to achieving climate neutrality and to implementing the principles of the European Green Deal.

The construction industry is becoming a key driver of 'green transition', which creates an urgent need for specialists with up-to-date knowledge in the field of Building Information Modelling (BIM), energy-efficient design, digital management, and sustainable development.

The BRIDGE project integrates the best practices of the European Union into the system of training Ukrainian engineers, architects, and construction professionals of a new generation. The main goal of the BRIDGE project is to bridge the gap between universities and industry by developing a new interdisciplinary Master's programme aimed at fostering digital and green competences in the construction sector.

To achieve this goal, the key objectives are:

- development and accreditation of Master's programmes in five Ukrainian universities
- modernization of BIM centres and creation of educational and methodological materials

- capacity building and professional development of teaching staff at leading European universities
- strengthening university–business cooperation to improve graduate employability
- digitalization of learning modules and creation of an interactive e-BRIDGE platform
- establishment of the BRIDGE Network, bringing together universities, industry, and public institutions

Project consortium. The project coordinator is the Ukrainian State University of Science and Technologies (USUST, Dnipro). The project partners include: from Ukraine - Lviv Polytechnic National University, Kyiv National University of Construction and Architecture, Odesa State Academy of Civil Engineering and Architecture, National University ‘Chernihiv Polytechnic’, public organization ‘Academy of Construction of Ukraine’, Institute of Professional Qualifications, and the Ministry of Education and Science of Ukraine; from the EU - Warsaw University of Technology (Poland), University of Sannio (Italy), RWTH Aachen University (Germany), Slovak University of Technology in Bratislava (Slovakia), and the Slovak Chamber of Civil Engineers.

Such partnership ensures international exchange of experience, innovation transfer, and the harmonization of educational standards in the field of sustainable construction and digitalization.

Under the BRIDGE project, a number of key results have been achieved, aimed at improving the quality of education, developing digital technologies, and strengthening cooperation between universities and industry. In particular, modern Master’s programmes adapted to labour market needs have been developed; BIM laboratories and learning hubs have been established to provide students with practical training based on real case studies from the construction sector. Special attention has been paid to the training of teachers and trainers in sustainable construction and digital modelling, enabling the integration of European approaches into the Ukrainian educational space. An important milestone was the creation of the interactive e-BRIDGE platform, which serves as a tool for international knowledge exchange, sharing of learning materials, and collaboration among project participants. At the same time, the ‘university–industry’ partnership network is being developed to align the educational process with real economic needs, supporting graduate employability. A Dissemination & Sustainability Plan has also been developed to ensure a long-term impact of the project on Ukraine’s higher education system.

The implementation of the BRIDGE project has a significant impact on all levels of the educational ecosystem. For students, it provides an opportunity to acquire modern competences in the fields of BIM technologies, energy efficiency, and sustainable development. For teachers, it offers a path toward professional growth, access to advanced European methodologies, and expanded academic mobility. For universities, it represents the modernization of education, increased international visibility, and stronger positions in global rankings. For the construction industry, it ensures the training of a new generation of professionals capable of working in the framework of green economy and digital transformation. For society, it creates the foundations for sustainable urban recovery, improved quality of life, and the country’s energy independence.

Thus, the BRIDGE project is an integral part of the European Union’s strategy to support Ukraine during the post-war recovery period. Its implementation contributes to the formation of a new culture of engineering education based on the principles of sustainable development, innovation, and partnership between academia and industry. The BRIDGE is not only an educational initiative but also an investment in human capital, technology, and the future of Ukraine - one that combines European values with the nation’s aspiration for modernization, renewal, and integration into the common European space.

Yuliia Bulhakova, Project Coordinator
Nataliia Chernova, Project Man

1. CASE STUDY: MODELING OF HISTORICAL BUILDINGS USING BIM TECHNOLOGIES BASED ON POINT CLOUDS

Hanieiev T. R.

INTRODUCTION

Prototyping historical buildings is the process of creating a digital model of an architectural object that reproduces its shape, structure, and sometimes functional features based on available data. Such a prototype can be highly accurate (based on LiDAR scans) or reconstructive (based on archival sources, descriptions, and photographs).

Prototypes can be used for the preservation of cultural heritage, virtual reconstruction of lost or damaged objects, scientific research, creation of interactive educational and museum exhibits, and preparation for restoration work (Figure 1).

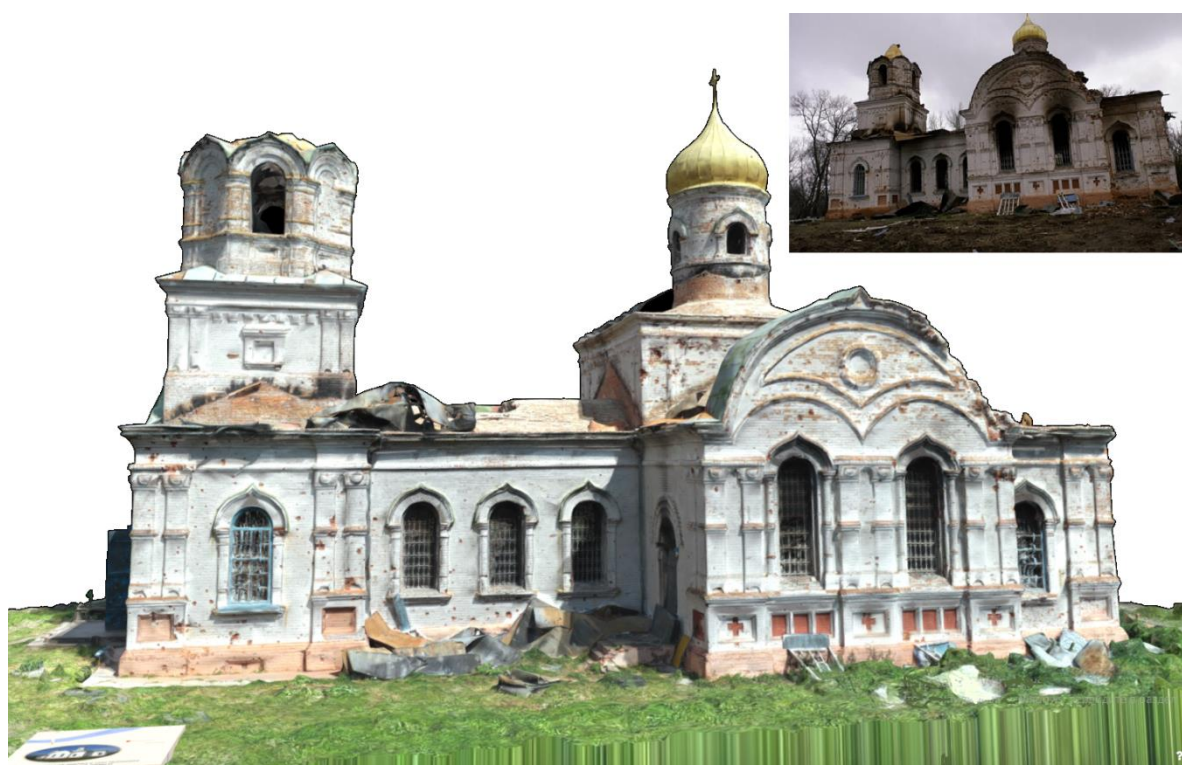


Fig. 1. Prototype of the Ascension Church in the village of Lukashivka, Chernihiv region

The value of a digital prototype lies in its ability to visualize a heritage object in its historical context, compare stages of building reconstruction, create an interactive model for VR/AR environments, and analyze structural features without physical intervention.

In modern practice of digital reconstruction of historical buildings, 3D scanning and BIM (Building Information Modeling) are closely interconnected. Together, they form a powerful toolkit for documenting the condition of an object, creating accurate digital models, planning restoration or conservation work, and integrating the object into the contemporary urban environment.

3D scanning is the process of collecting spatial data about an object using one of the three most common technologies: LiDAR scanning, photogrammetry, or structured light scanning.

The advantages of scanning over manual measurements include high accuracy, the ability to capture complex shapes and decorative elements, and the possibility of working with objects in an emergency or unstable condition.

The result of 3D scanning is a point cloud a set of coordinates describing the surface of the object, which is later transformed into a mesh model or a BIM object.

BIM, in the context of historical buildings, allows for the integration of archival sources (plans, drawings, photos), facilitates coordination between architects, restorers, and engineers, can include versions of the object from different historical periods, and supports digital documentation of changes (restorations, losses, reconstructions).

PURPOSE AND OBJECTIVES OF THE STUDY

The purpose of this study is to enhance the professional skills of learners in addressing multifaceted historical, cultural, and architectural-restoration issues related to the digitalization of historical heritage.

The main objective of the study is to create a digital three-dimensional prototype of a historical building based on LiDAR data, available archival sources, and modern 3D modeling methods, followed by an analysis of its architectural features, historical transformations, and preservation status.

Specific objectives of the study include:

1. Mastering methods for working with point clouds obtained through LiDAR scanning;
2. Integrating archival materials (photographs, drawings, descriptions) into the modeling process;
3. Constructing a 3D model of the object with consideration of historical accuracy;
4. Using software for geodata processing and creating BIM or mesh models;
5. Analyzing changes in the structure and appearance of the object across different historical periods;
6. Preparing graphical materials and a report reflecting the research results;
7. Deepening teamwork skills and collective decision-making.

After creating a digital prototype of a historical building, it is important not only to visualize the object but also to analyze it in terms of accuracy, historical authenticity, structural features, and changes over time (Figure 2).

Reliable LiDAR data acquisition is ensured by using a scanner such as the FJD Trion P1 LiDAR a powerful, portable tool for 3D scanning, ideally suited for documenting historical buildings due to its technical specifications and ease of use:

Accuracy: up to 2 cm at a scanning distance of up to 40 meters;

Field of view: $360^{\circ} \times 59^{\circ}$;

Scanning speed: 200,000 points/sec;

Portability: weighs approximately 1 kg

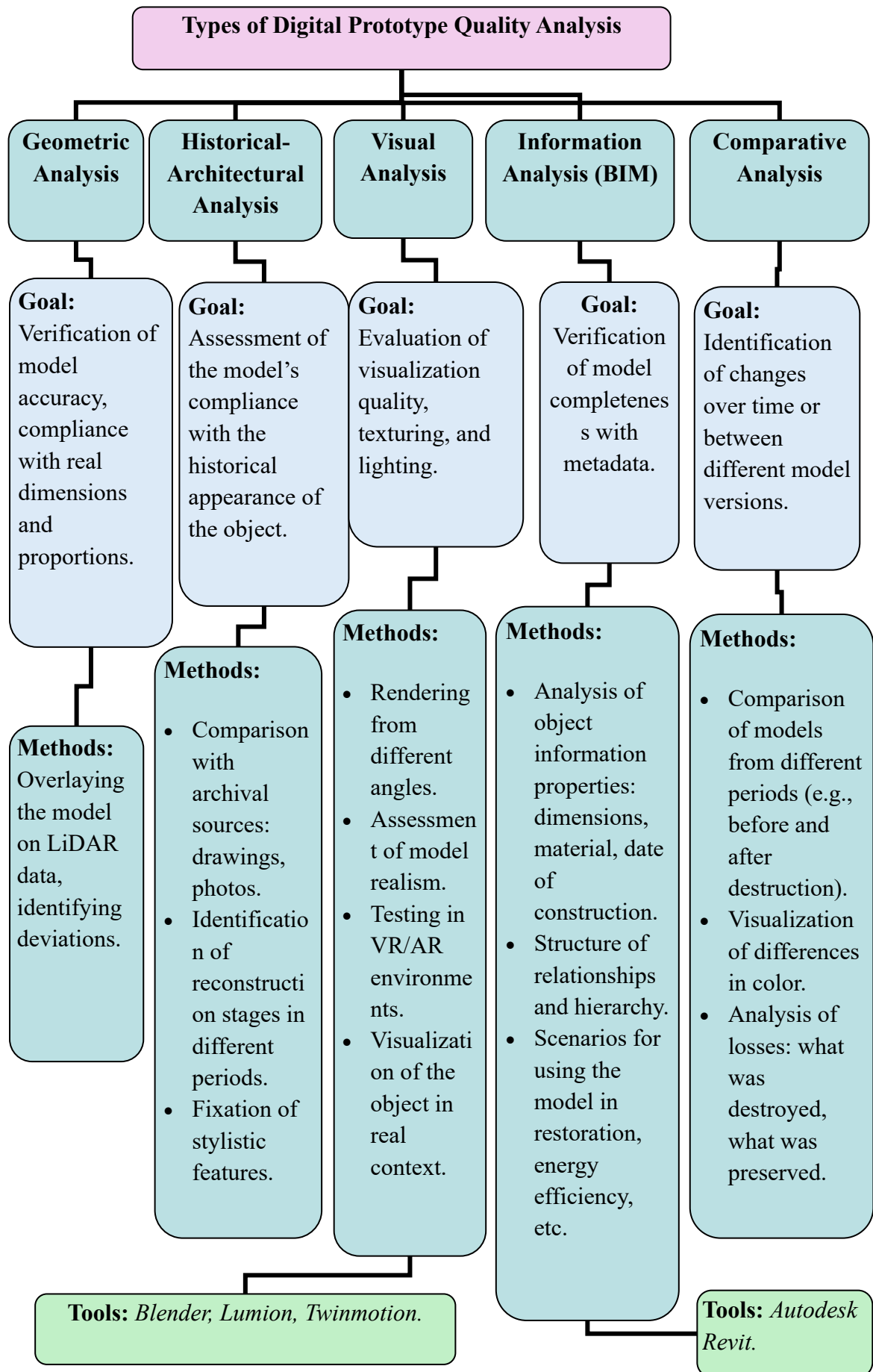


Fig. 2. Types of Digital Prototype Quality Analysis

If technically required, scanners from other manufacturers and alternative data processing technologies may be used.

SCANNING METHODOLOGY

Main Scanner Specifications

The FJD Trion P1 SLAM scanner (Fig. 3) is a portable laser 3D scanner with SLAM (Simultaneous Localization and Mapping) technology support, which enables real-time scanning without the use of targets or a tripod. It is designed for mobility and speed, with automatic frame alignment, which makes its data processing logic different from traditional stationary LiDAR solutions.



Figure 3 – FJD Trion P1 Scanner. General View

The scanner is controlled via a smartphone application; all operations during scanning and visual monitoring are performed through the mobile app interface. After data acquisition and saving, the files are exported to a PC/laptop for further processing.

This scanner uses LiDAR + IMU (Inertial Measurement Unit) technology. The maximum object capture range is 120 m, but this assumes ideal scanning conditions with 0% surface reflectivity and no obstacles. In practice, the optimal data capture range is 50–60 m. After scanning, the point cloud can be processed in the following formats: .las, .ply, .xyz, .pcd, .e57 (detailed description will follow later).

Preparation for Scanning

Before starting work with the scanner, ensure the following:

- Sufficient battery charge;
- Battery securely attached to the scanner unit;
- Camera securely attached to the scanner unit;
- LiDAR camera lens is clean;
- Stable connection to the control smartphone.

Device Startup (Fig. 4):

1. Place the P1 on a flat surface;
2. Press and hold the power button for 1–2 seconds to turn on the P1. The status indicator will slowly blink green;
3. Wait about 15 seconds until the status indicator turns solid green. Initialization is complete, and the P1 enters standby mode.

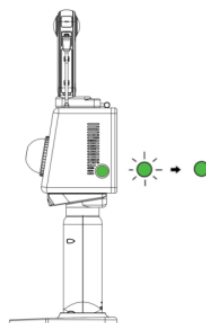


Figure 4 – FJD Trion P1. Visual Interface

Wi-Fi Connection:

1. Turn off cellular service on your phone, enable Wi-Fi, and then find the Wi-Fi network named after the device's serial number (located on the back panel of the P1).
2. Enter the password "*****" to connect to the network.
3. Open a browser on your phone and enter "*****" in the address bar to access Trio Scan (Fig. 5).



Figure 5 – Main Window of the FJD Trion Scan Application

Data Collection:

1. Tap “Scan” in Trion Scan, enter the project name, and then tap “OK”. The scanning initialization will begin, and the power button status indicator will blink blue. The P1 must remain stationary during this process (Fig. 6).
2. Wait approximately 15 seconds until the power button status indicator starts blinking red.

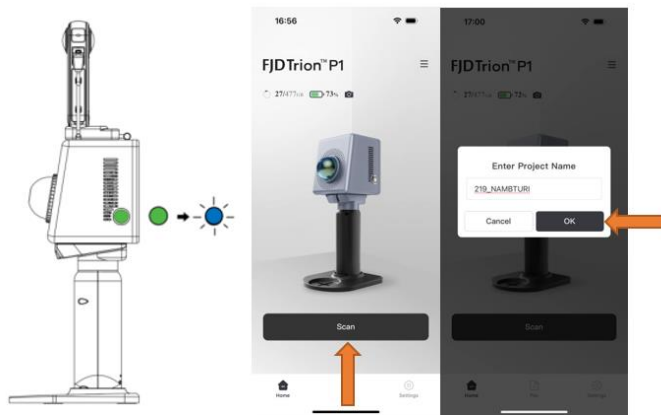


Figure 6 – Start of Scanning in FJD Trion Scan

The P1 is now ready for scanning, and you can begin the process while monitoring the point cloud data in Trion Scan (Fig. 7).

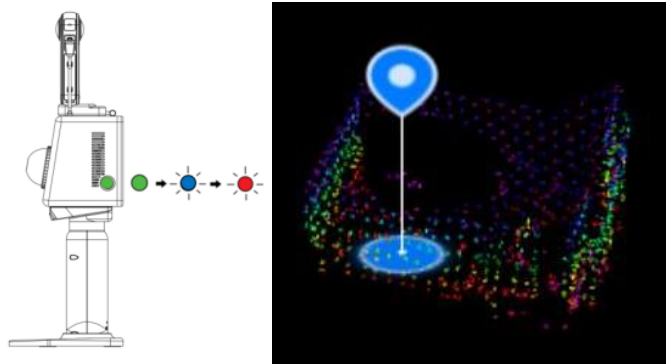


Figure 7 – Scanning Process in FJD Trion Scan

Tap “Finish”, then “OK” to save the data. The power button status indicator will blink blue twice.

Scanning Procedure

The scanning route should cover all blind spots of the object. The number of passes over the same area does not negatively affect scan quality, so it can be ignored. To capture more data points, it is recommended to plan the route in concentric loops around the object, gradually increasing the distance after each loop. The route should avoid sharp elevation changes, and the presence of people in the scanning area should be minimized.

The movement speed during scanning should not exceed a normal walking pace. The longer the scanner remains at a point, the more data it captures (however, stopping to increase point density is unnecessary).

During scanning, monitor the point cloud generation in the app interface. If overlaps, loss of spatial orientation, scanner disconnection, or other issues occur, stop the process and restart scanning.

Completion and Data Export

At the end of the scanning route, close the loop by retracing 5–10 meters of the path to improve point cloud alignment. After scanning, disassemble the scanner and store it in the case.

To export point cloud files, connect the scanner to a PC/laptop via Wi-Fi with the FJD Trion Model software installed.

To download data:

- Tap “Data” at the bottom,
- Tap “Download” for the required dataset,
- Select the desired format in the pop-up window, then tap “Download” (Fig. 8).

Export files in the following formats:

- .fjdslam – for point cloud processing in FJD Trion Model,
- .las – for point cloud processing in Autodesk ReCap,
- .insv – for applying color textures (video file from the camera).

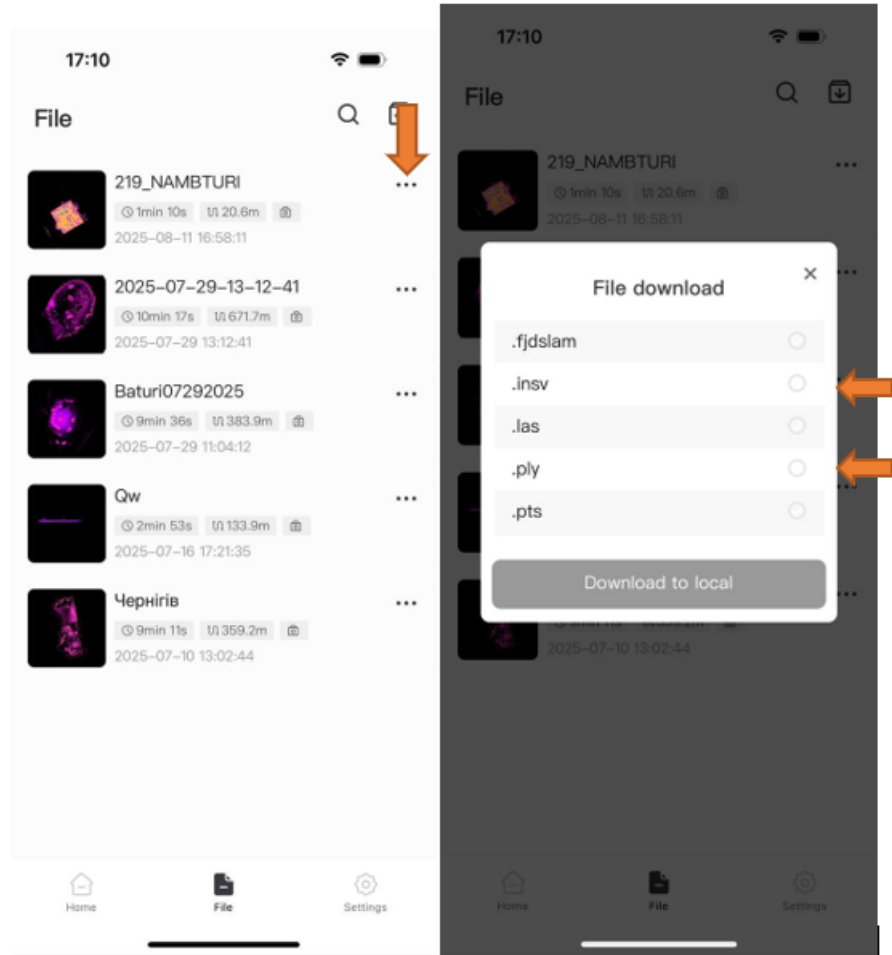


Figure 8 – Downloading Files from the Application

After completing the point cloud registration process in FJD Trion Model, you obtain a preliminary structure of the scanned object's point cloud with generated geometry and applied textures (Fig. 9).

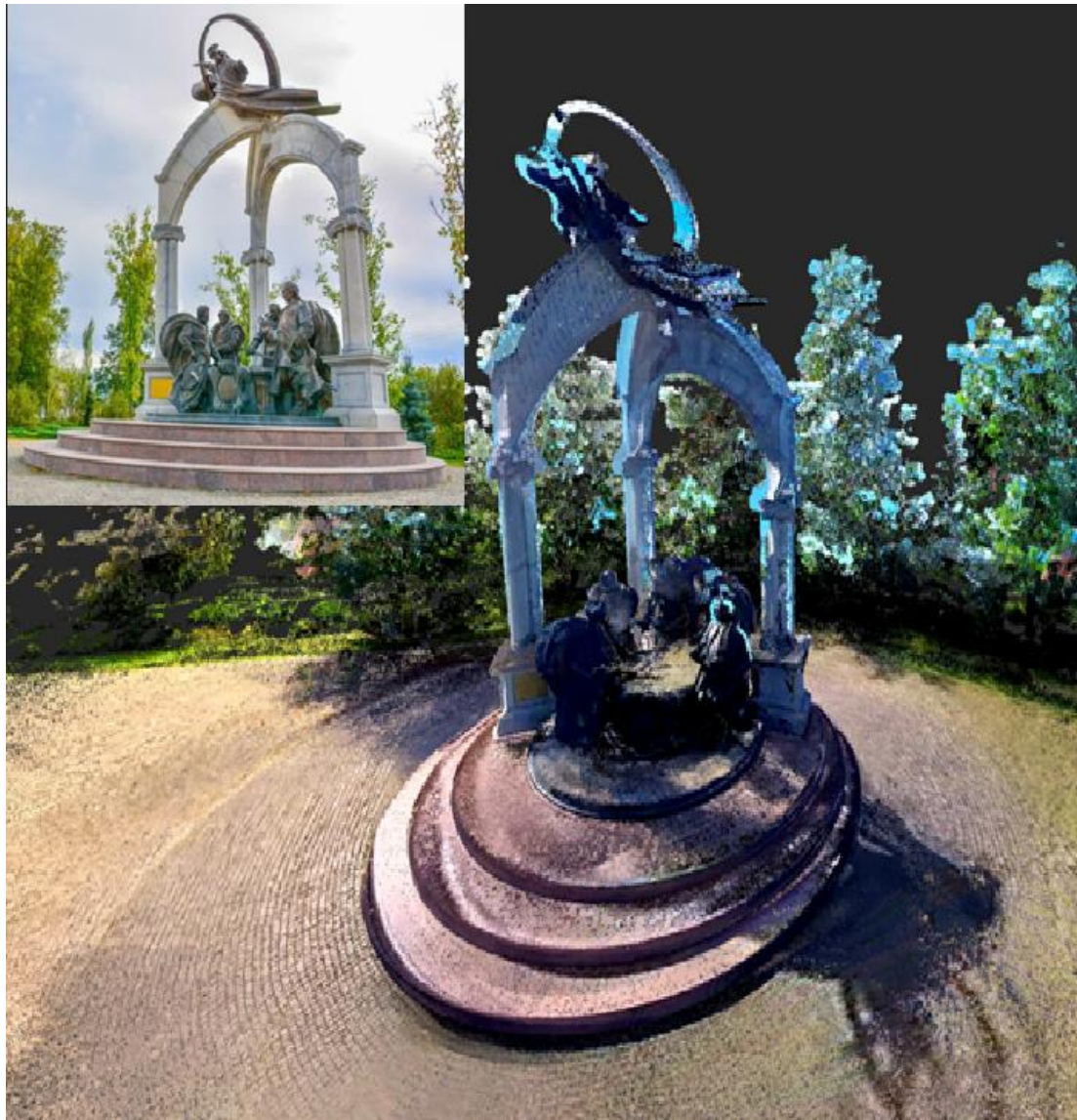


Figure 9 – Results of Scan File Processing in FJD Trion Model

After completing the point cloud registration process in FJD Trion Model, export the file in .e57 format for further processing in Autodesk ReCap.

Adding the Point Cloud to a Revit BIM Project

To process the point cloud in Revit, first link the point cloud file to your project using the Insert → Link → Point Cloud (Fig. 10). The .rcp file format is a project file that groups multiple .rcs scan files. The result of indexing the original file format is an .rcp file and one or more .rcs files. In the File Name field, select the file or specify its name.

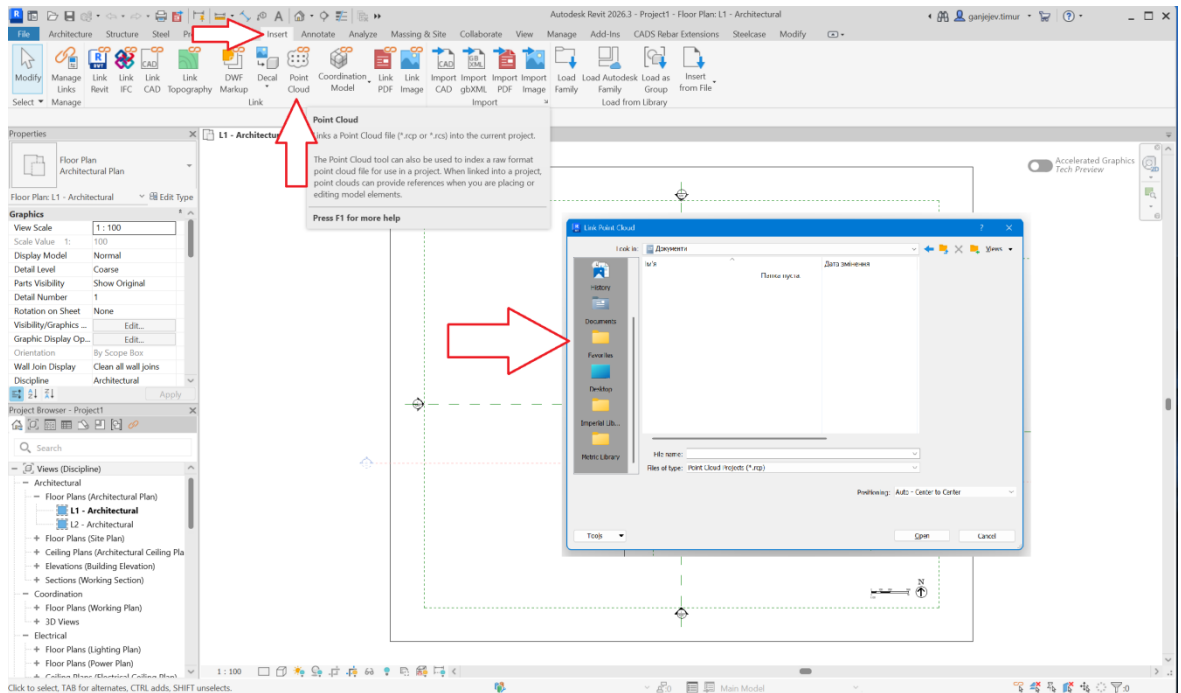


Figure 10 – Adding a Point Cloud to a Revit BIM Project

For the Positioning parameter, choose one of the following options (Fig. 11):

- **Auto – Center to Center**

Revit places the center of the point cloud's bounding box at the center of the model's bounding box. If the model does not fit in the current view, its center may not be visible. In this case, zoom to fit the model in the window. The Revit model will then be centered in the view.

- **Auto – Origin to Internal Origin**

Revit places the origin of the point cloud (e.g., point (0,0,0)) at the Revit project base point, which can be considered the project's reference point on the site plan. When rotating **Project North**, Revit rotates the point cloud so that its north direction (0,1,0) aligns with the current **Project North**. This option is recommended if the point cloud was created with a known reference point and direction in the model or on-site.

Note: When using **Origin to Origin** for a point cloud stored with geographic/geodetic coordinates, the cloud may appear far from the rest of the model.

- **Auto – By Shared Coordinates**

Revit assumes that the point cloud file uses the same shared coordinate system as the model. The point cloud origin will be placed at the shared coordinate origin, accessible via the survey point. The point cloud will be oriented so that its north direction (0,1,0) matches the true north of the Revit model.

- **Auto – Origin to Last Placed**

Revit aligns the next imported point cloud with the previously imported one. This option becomes active after inserting the first point project. You can move the first cloud to align it with model elements. If additional point clouds were created on the same site and in the same system as the first one, this option ensures correct relative placement of new clouds.

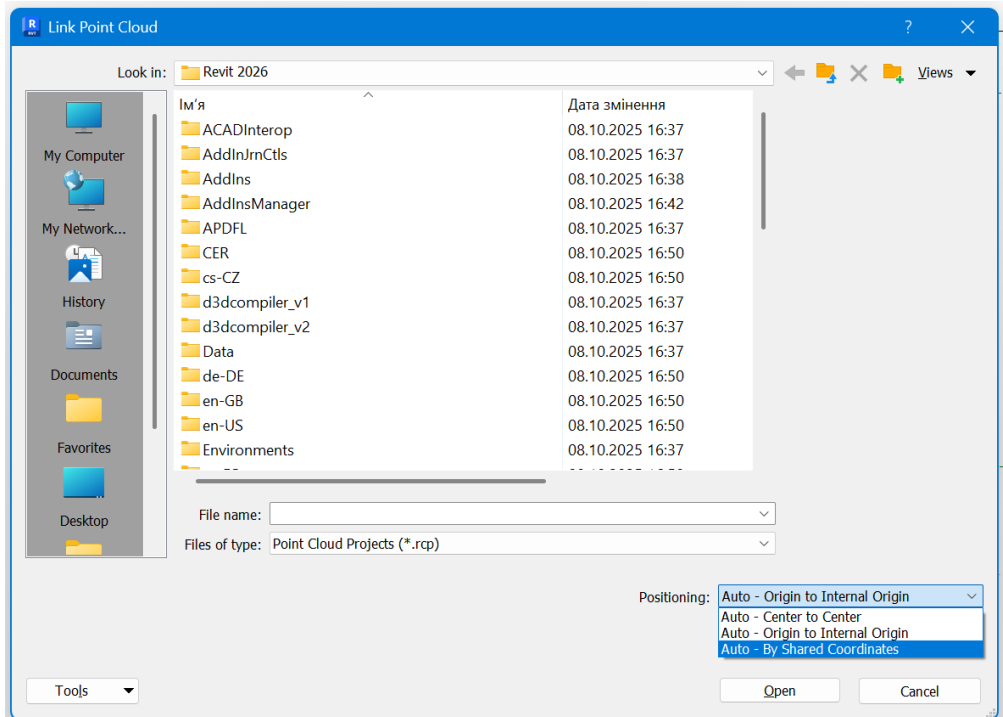


Figure 11 – Point Cloud Positioning Options

After linking, you can manage the visibility of the point cloud through the Visibility/Graphics Overrides dialog box, as well as create geometry using the snap-to-plane function or directly to point cloud points.

In this dialog box, you can enable or disable point cloud visibility for a specific view and change the color display mode of the points.

Creating Geometry Based on the Point Cloud

Use Revit tools to create geometry (e.g., Wall, Line, Grid). The snapping function simplifies model creation by referencing point cloud data. In some cases, it is useful to enable automatic detection of implicit planar surfaces in the point cloud and allow snapping to them. If you need to snap directly to a point in the cloud, press Tab to cycle through snap options if the plane snap is activated first.

Order of Describing Research Results

The research object is selected by the student based on literature review and approved by the instructor. Research can be conducted in groups of 2–3 students. Before scanning, perform thorough photo documentation of the object's preservation state, followed by measurements of the main façade elements.

The report (Appendix 1) should include:

- A brief historical overview;
- Description of the current state with photos;
- Analysis of results in terms of accuracy, historical authenticity, structural features, and changes over time.

Drawings of the scanned object should include:

- Façades generated from the point cloud in Revit (sample in Appendix 2), A4 format, 1–2 sheets.
- Axonometry and characteristic fragments of façade decoration, windows, doors, etc., A4 format, 1 sheet (Appendix 3).
- If previous research data exists (plans and sections differing from the current state), they can be compiled on an additional sheet with author names and research year indicated (Appendix 4).

Appendices

Appendix 1.

20

Опис

Комплекс Михайлівської церкви є прикладом архітектури зрілого класицизму в Україні кінця XVIII століття. Комплекс включає Михайлівський храм, оточений кам'яною огорожею із дзвіницею та вежами. Зберігся у спокійному вигляді, крім огорожі, яка є тільки з двох (західної та південної) сторін, у розі розташовується вежа.

Споруджена в період 1782—1796 роки над річкою Трубіж.

Цегляна, оштукатурена, хрестоподібна у плані церква з невеликою апсидою та кутовими камерами на всю висоту, одностовпа. Храм вінчає напівсферичний купол на круглomu барабані, чотири малі декоративні гавки над кутовими камерами. Фасади по центру мають невеликі ризаліти зі складними (чотири спарені колони підтримують арку) доричного ордера портиками з трикутними фронтонами. До одного з портиків примикає напівкругла з напівколонна апсида (ніша), де розміщується вівтар.

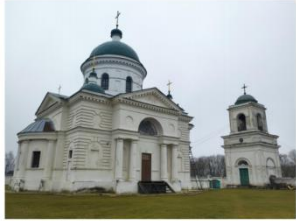


Рисунок 14 – Михайлівська церква та дзвіниця

7




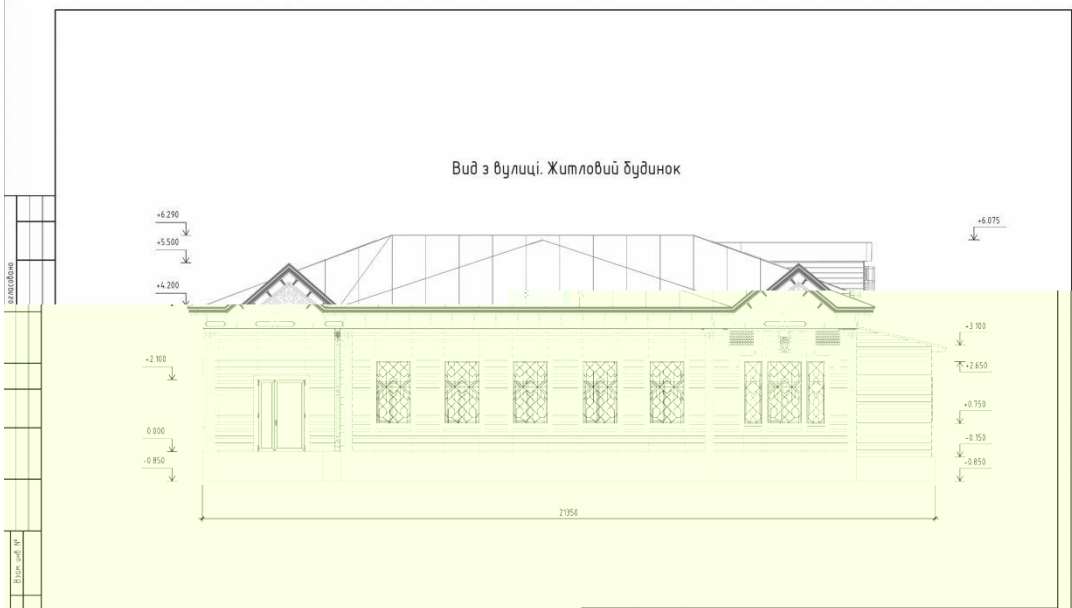
Рисунок 4 – Фотофіксація фасадів

1 – Фото А.1.1;	6 – Фото А.1.18;	10 – Фото А.1.25;
2 – Фото А.1.6;	7 – Фото А.1.19;	11 – Фото А.1.30;
3 – Фото А.1.9;	8 – Фото А.1.23;	12 – Фото А.1.31;
4 – Фото А.1.11;	9 – Фото А.1.24;	13 – Фото А.1.32;
5 – Фото А.1.17;		

Фото дивись додаток А.

Appendix 2.

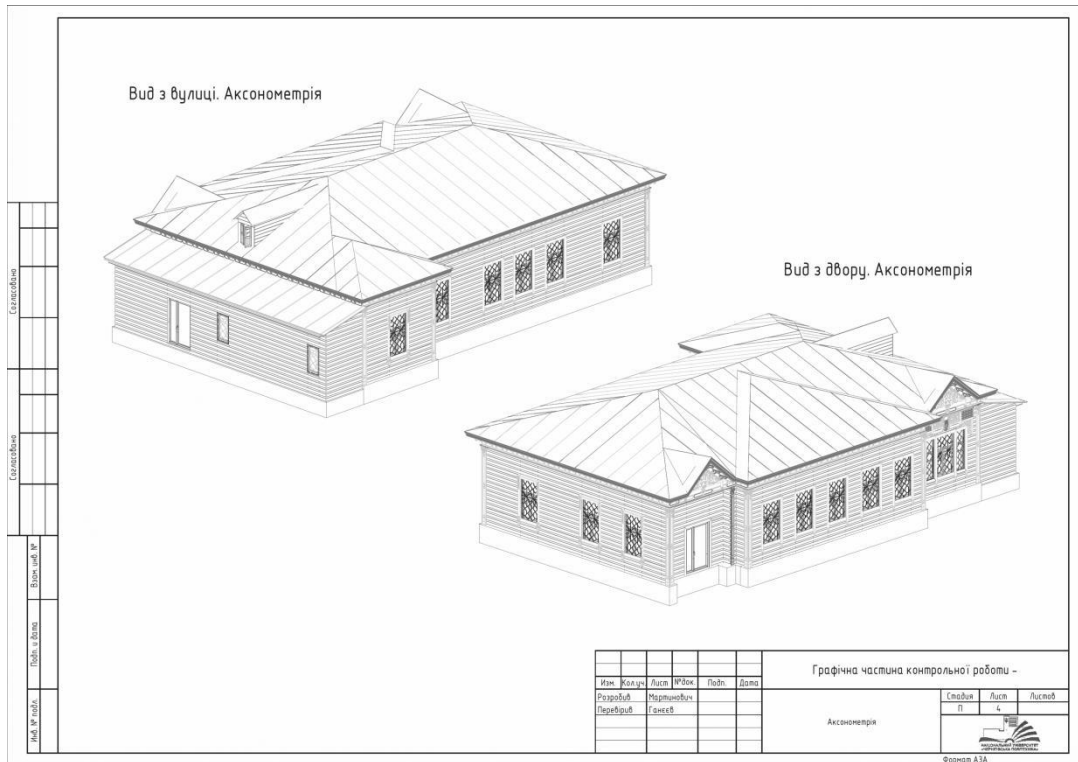
Вид з вулиці. Житловий будинок



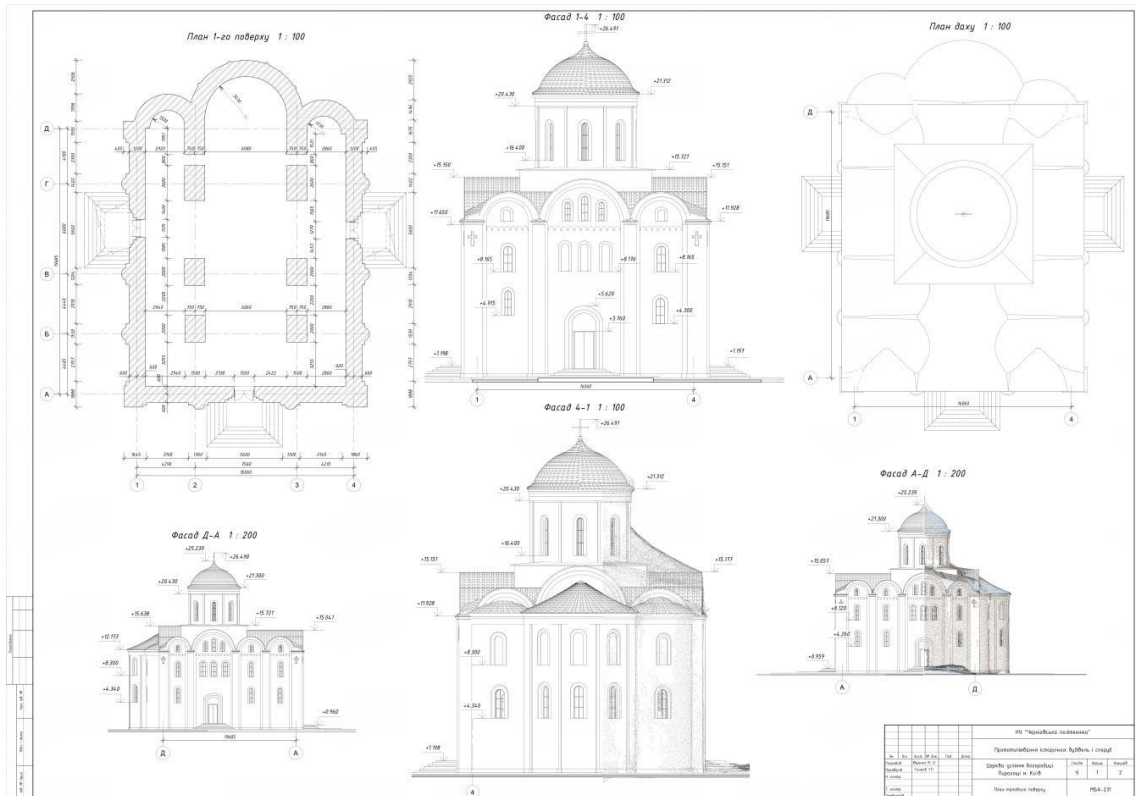
Графічна частина контрольної роботи –					
Чернетів вул.Ганець, 92					
Ім'я	Колір	Лист	Місця	Підпис	Дата
Розробив			Мартинюк		
Перевірив			Ганець		
Житловий будинок 19 ст.					Старий
Вид з вулиці Житловий будинок					Лист
					1
					Листів
					1

Формат А3

Appendix 3.



Appendix 4.



References

1. Using Point Cloud Files in a Project. Autodesk Revit 2014. URL: <https://help.autodesk.com/view/RVT/2014/RUS/?guid=GUID-D179BB6C-5528-498F-9413-00237092C2FA> (date of access: 16.05.2025).

2. KORZACHENKO, M., HANIEIEV, T., & PETRENKO, I. (2025). Digital technologies in the implementation of restoration works of underground complexes. *Bases and Foundations*, (50), 113–122.
3. PETRENKO, I., et al. Photogrammetry in the preservation of cultural and historical heritage. Methodology and perspectives. 2024.
4. PETRENKO, I. O.; HANIEIEV, T. R. (2021). Determination of the error of an optical scanner.
5. DANESHMAND, Morteza, et al. 3d scanning: A comprehensive survey. arXiv preprint arXiv:1801.08863, 2018.
6. HELLE, Robin H.; LEMU, Hirpa G. A case study on use of 3D scanning for reverse engineering and quality control. *Materials Today: Proceedings*, 2021, 45: 5255-5262.
7. JEDLIŃSKI, Maciej, et al. 3D scanners in orthodontics—Current knowledge and future perspectives—A systematic review. *International journal of environmental research and public health*, 2021, 18.3: 1121.
8. PIEKARCZUK, Artur, et al. A Case Study of 3D Scanning Techniques in Civil Engineering Using the Terrestrial Laser Scanning Technique. *Buildings*, 2024, 14.12: 3703.
9. RIBEIRO, Nuno Ferrete; SANTOS, Cristina P. Inertial measurement units: A brief state of the art on gait analysis. In: 2017 IEEE 5th Portuguese Meeting on Bioengineering (ENBENG). IEEE, 2017. p. 1-4.
10. SAMATAS, Gerasimos G.; PACHIDIS, Theodore P. Inertial measurement units (imus) in mobile robots over the last five years: A review. *Designs*, 2022, 6.1: 17.
11. VADUVESCU, Vlad Aurelian; NEGREA, Petre. Inertial Measurement Unit—A Short Overview of the Evolving Trend for Miniaturization and Hardware Structures. In: 2021 International Conference on Applied and Theoretical Electricity (ICATE). IEEE, 2021. p. 1-5.
12. BAKER, Wayman E., et al. Lidar-measured wind profiles: The missing link in the global observing system. *Bulletin of the American Meteorological Society*, 2014, 95.4: 543-564.
13. WULDER, Michael A., et al. Lidar sampling for large-area forest characterization: A review. *Remote sensing of environment*, 2012, 121: 196-209.
14. BEHROOZPOUR, Behnam, et al. Lidar system architectures and circuits. *IEEE Communications Magazine*, 2017, 55.10: 135-142.
15. ALTUNTAS, Cihan. Review of scanning and pixel array-based lidar point-cloud measurement techniques to capture 3D shape or motion. *Applied sciences*, 2023, 13.11: 6488.
16. TRIGLAV-ČEKADA, Mihaela; CROSILLA, Fabio; KOSMATIN-FRAS, Mojca. A simplified analytical model for a-priori LiDAR point-positioning error estimation and a review of LiDAR error sources. *Photogrammetric Engineering & Remote Sensing*, 2009, 75.12: 1425-1439.
17. AULINAS, Josep, et al. The SLAM problem: a survey. *Artificial Intelligence Research and Development*, 2008, 363-371.
18. MACARIO BARROS, Andréa, et al. A comprehensive survey of visual slam algorithms. *Robotics*, 2022, 11.1: 24.

2. THERMAL EFFICIENCY ASSESMENT OF ECO-ORIENTED PREFABRICATED MODULAR BUILDINGS: CASE STUDY

Bolotov M. H., Rudenko M. M.

Abstract

The main purpose of this study was to provide a deeper understanding of thermal performances of eco-oriented prefabricated modular housing based on the real example of a frame modular single-family building "QHome-26", located on the outskirts of the Chernihiv city.

By using infrared thermography, monitoring of existing cold bridges and the main heat losses in the building's envelope was carried out as the first and useful step for a good energy audit. For deeper understanding of building's envelope energy performance, the meticulous thermal engineering calculation was done.

The next step was to propose possible redevelopment of a structural and energy plant that promotes the building energy rating higher.

Keywords: *Sustainability, modular buildings, eco-construction, thermal efficiency.*

1. Introduction

The relevance of eco-oriented modular construction is constantly growing due to the aggravation of housing and environmental problems associated with military operations on the territory of Ukraine. Modular prefabrications or modular prefabs implies the off-site manufacturing where whole building or its separate components are manufactured in controlled factory environment with further transportation and installation on construction site [2].

The intentions to go green and minimize of natural resources consumption requires shifting of our habitual construction practices toward the more sustainable and energy-efficient ones [1]. In this context, the modularization in construction becomes one of the modern trends. This is mainly due to the fact that modular prefabrications focus directly on the ecological and energy saving approaches in building construction that allows to downsize hazardous environmental impact, resources and energy consumption and construction waste.

2. The statement of basic materials

Our case study is the single-family modular prefab "QHome-26" [3], which was off-site manufactured in 2023 by the Ukrainian company "QHome" supported by the UN Refugee Agency (UNHCR) and installed in the vicinity of Chernihiv city in Novoselivka village. As a foundation of the building, the FBS base blocks were used. The building's exterior, its installation process, 3D model of the timber carcass as well as the premises plan with its explication are given in Fig.1.



Fig. 1. External view of studied single-family modular prefab



Fig. 2. Installation process of single-family modular prefab



Fig. 3. 3D model of the timber carcass

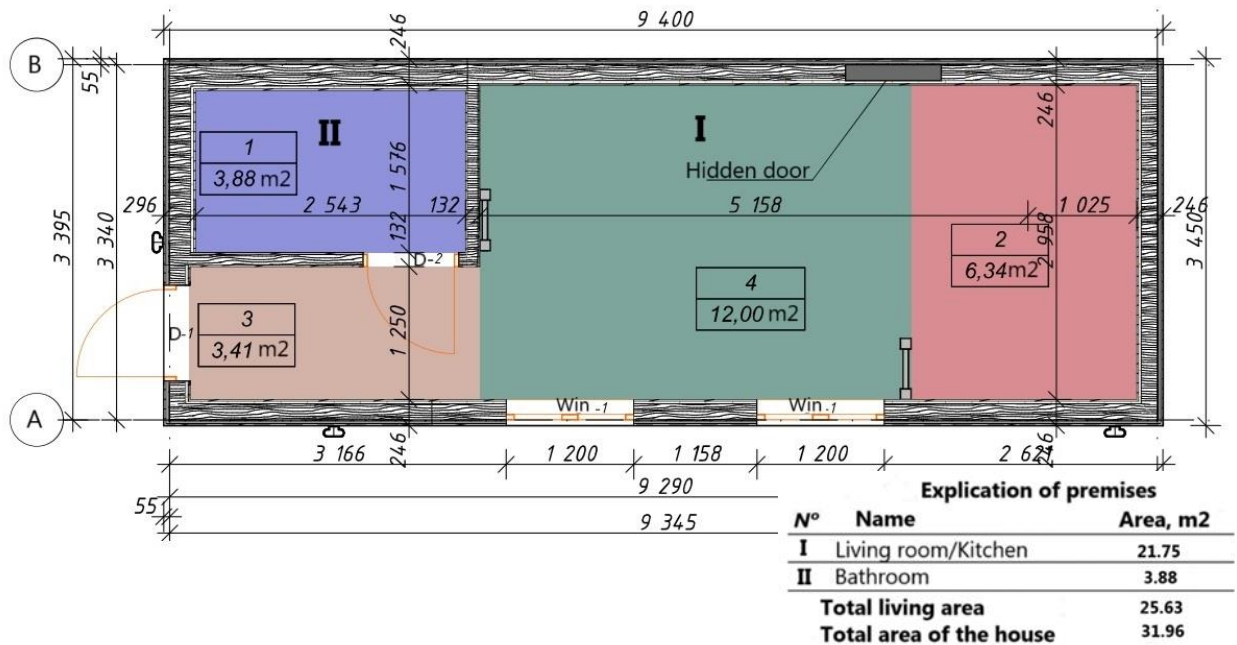


Fig. 4. The plan of the premises with its explication

The basis of the house is a timber-made frame with the dimensions of 9400×3450 mm and the thickness of the partitions of 300 mm. The building is made up of two premises, particularly, combined living room and kitchen with the area of 21.75 m² and the bathroom of 3.88 m². Thus, the living area of 25.63 m² and the total area of the building of 31.96 m².

As far as it's known, the building's envelope plays a crucial role in terms of thermal performances of the buildings especially when it comes to the modular ones. In this case, the core

content of the building's energy efficiency and sustainability is the thermal insulation performance of enclosure structures.

In our case study, as the insulator the ecologically friendly basalt wool was used which provides a high level of thermal efficiency with the low coefficient of heat conductivity about 0.038 W/m·K. On top of that, the basalt wool well-known as non-combustible, vapor permeable and soundproofing one.

The enclosing structures were manufactured as the sandwiches, which are made up of the timber frame, hydro barrier membrane, OSB plates, insulator, vapor barrier, and wooden lining as an internal decorative layer. The roofing consists of two layers of felt paper, OSB, timber checkrails, hydro barrier membrane, insulated timber rafter, vapor barrier, checkrails and wooden lining. The floor is detached from the ground on a distance of 250 mm and made up of the timber checkrails, two layers of felt paper, OSB, timber beams, insulator, OSB and PVC internal lining.

Reinforced timber frame of the building is the dry calibrated timber with the OSB-3 plates, bitumen primer, vapor and waterproof, roofing material (roofing felt EPP 2.5 and EPP 4.0.). The construction details of the enclosures (walls, floor, and the roof) are given in Fig. 5. It should be noted that the thickness of the walls and roof is 250 mm while the thickness of the floor is 300 mm.

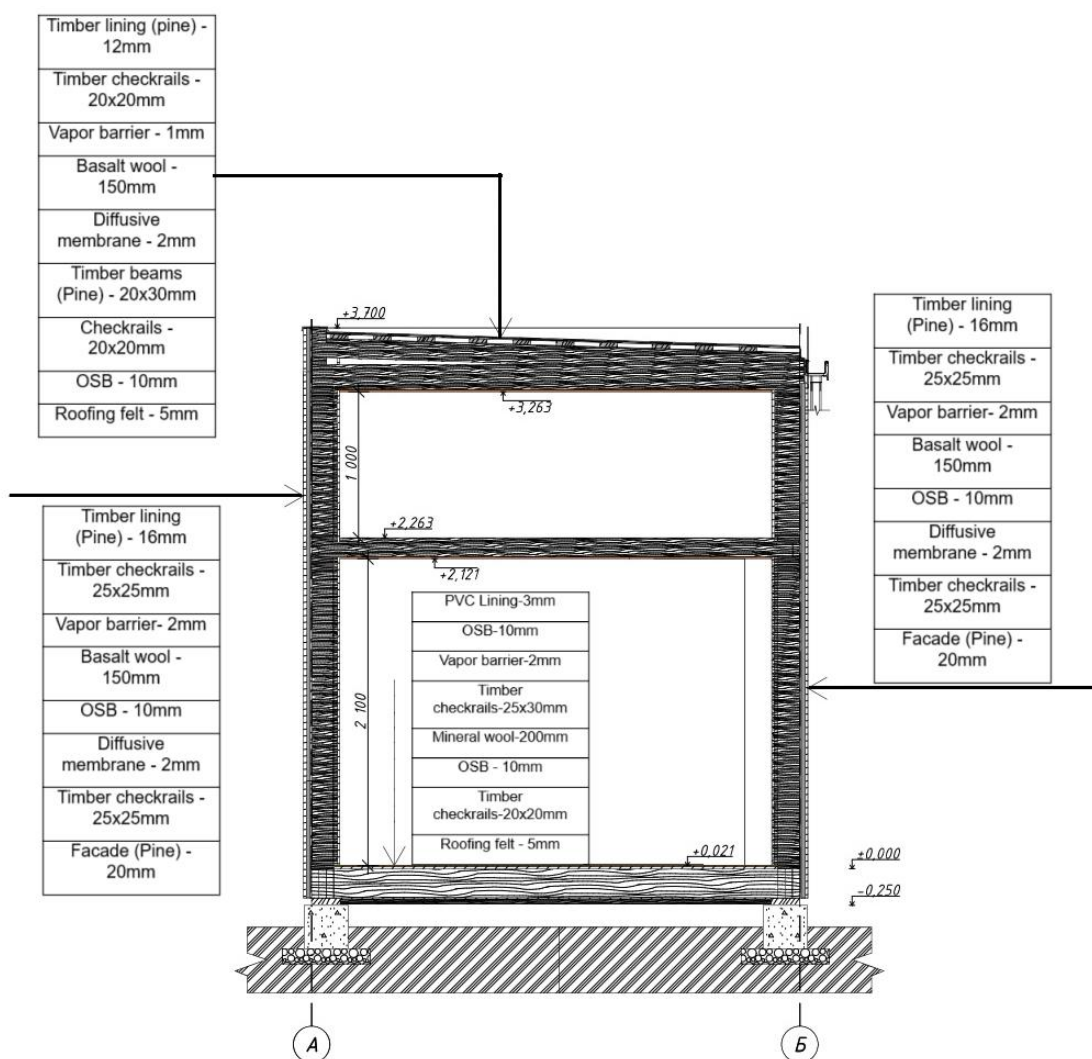


Fig. 5. The detailed content of the enclosures of studied modular building

On the Southern facade, there are two triple-glazed windows of 4-10-4-10-4 type with the three air filled chambers profile system with the dimensions of 1.2×1.2 m were installed.

The house is heated up by the solid fuel boiler with the power of 7 kW, which is easily allows to heat the building up to the total area of 100 m².

Before the assessment of studied building's thermal efficiency the infrared thermography inspection in order to get the information about the existing thermal bridges and the main heat loss in the building envelope was conducted.

3. The infrared thermography inspection

The inspections were carried out on 12th of February 2025 approximately from 10:00 to 11:00 am when the weather was overcast, the temperature of the environment was about - 16 °C, wind speed doesn't exceed of 3 m/s with the humidity level of 60 %. The probability of precipitations was about 10 %. It should be noted that inside temperature at the time of inspection was fixed at a level of 21 °C more the 12 hours. The inspections of thermal bridges were carried out with HTi thermal imaging camera.

The inspection methodology consists of the following steps:

1. Checking the total heat loss through the walls, windows and roof.
2. Searching for areas with increased heat loss in typical problem zones.
3. Analyzing heat dissipation through gaps in window sashes and doorways.
4. Assessing heat leaks through ventilation and air conditioning systems (if available).

The results of infrared thermography are given in Fig. 6.

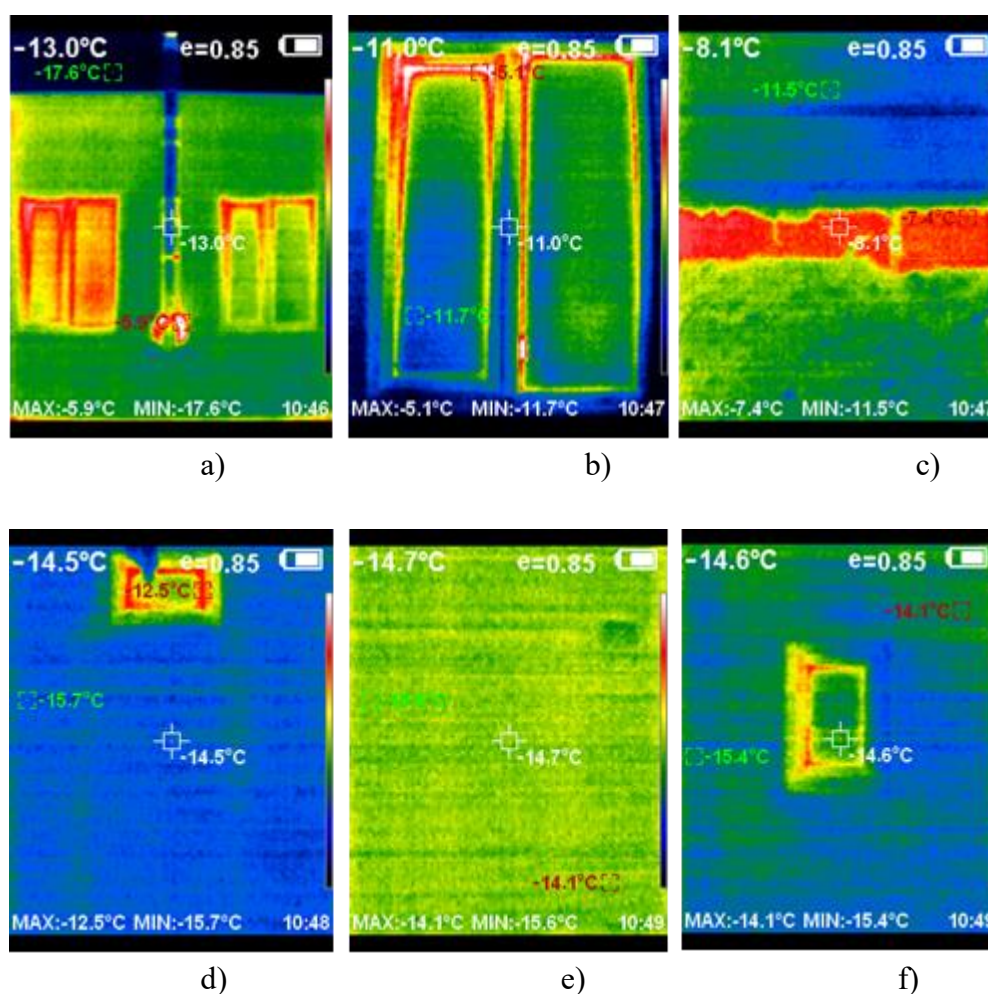


Fig. 6. Thermogram of Southern facade (a), window (b), basement (c), eastern facade (d), Western facade (e) and Northern facade (f) of inspected building

Analyzing of obtained infrared thermal images of studied building it can be noticed that the main weaknesses that affect negatively of enclosure's thermal behavior is the windows. The thermal bridges concentrated directly along the window's frame contour, which can evidence both the poor quality of installation, as well as the low quality of manufactured frame and insulating glass. It can be seen that on the surface of one window the temperature varies from -11 up to -5 °C depending on the measurement point, which indicates the heterogeneity of the heat flow passing through the window and the heterogeneity of the thermal resistances of its components accordingly. From the other hand, as it was already mentioned, the off-site manufacturing excludes the fabrication errors because of meticulous quality control of all production stages.

The other significant aspect of heat loss that weakens thermal performances of examined building this is foundation. Remind that our case study contains the FBS base blocks as a foundation without external thermal insulation frames. Additionally, there are no any heat leakages through the heat exchanger as well as the split air conditioning system were detected.

4. Thermal engineering calculation

The calculation is carried out in order to determine of suitability of actual state of the enclosure structures thermal insulation to the regulatory requirements. Normally, it begins with determination of thermal characteristics of building's envelope. The material properties of the walls, floor and roofing are given in pivot Table 1.

Table 1 – Material characteristics of enclosure structures

of layer	Material	Thickn	Therma	Ref.	Thermal
		ess	l conductivity		resistance
		δ, m	$\lambda, W/(m \cdot K)$		$m^2 \cdot K/W$
External walls characteristics					
	Timber lining (Pine)	0.016	0.18	[4]	0.089
2*	Closed air gap	0.025	–	[4]	0.155
	Vapor barrier	0,002	0,3	[4]	0.0066
	Basalt wool	0.15	0.038	[4]	3.947
	OSB	0.01	0.13	[4]	0.077
	Diffusive membrane	0.002	0.23	[4]	0.0087
	Closed air gap	0.025	–	[4]	0.155
	Facade (Pine)	0.02	0.18	[4]	0.111
Characteristics of the floor					
	PVC Lining	0.003	0.33	[4]	0.009
	OSB	0.010	0.13	[4]	0.077

3*	Closed air gap	0.03	–	[4]	0.155
	Vapor barrier	0,002	0,3	[4]	0.006
	Mineral wool	0.2	0.044	[4]	4.762
	OSB	0.01	0.13	[4]	0.077
7*	Closed air gap	0.02	–	[4]	0.155
	Check rail (Pine)	0.03	0.18	[4]	0.167
	Roofing felt	0.005	0.17	[4]	0.029

Table 1 – Material characteristics of enclosure structures

of layer	Material	Thickn ess	Therma l conductivity	Ref.	Thermal resistance
		δ , m	λ , W/(m·K)		$m^2 \cdot K/W$
Characteristics of the roof					
	Timber lining (pine)	0.012	0.18	[4]	0.067
2*	Closed air gap	0.02	–	[4]	0.155
	Vapor barrier	0,001	0,3	[4]	0,003
	Basalt wool	0.15	0.038	[4]	3.947
	Diffusive membrane	0.002	0.23	[4]	0,0086
	Timber beams (Pine)	0.03	0.18	[4]	0.167
7*	Closed air gap	0.02	–	[4]	0.155
	OSB	0.01	0.13	[4]	0.077
	Roofing felt	0.005	0.17	[4]	0.029

*The systems of closed air gaps with the thickness of 0.025-0.03 m were provided by timber checkrails frame

The calculation of enclosing structures was conducted with accordance to the models are shown in Fig. 7.

Thermal resistance of every separate enclosing layer in table 1 were determined as follows [5]:

$$R_i = \frac{\delta_i}{\lambda_i}; \quad (1)$$

where R_i – thermal resistance of i layer of enclosure, ($\text{m}^2 \cdot \text{K}/\text{W}$); δ_i – the thickness of i layer; λ_i – thermal conductivity of i layer.

Thermal resistance of multilayered enclosing structure (R_0) can be determined as [5]:

$$R_0 = R_{\text{int}} + \sum_{i=1}^n R_i + R_{\text{ext}} = R_{\text{int}} + \sum_{i=1}^n \frac{\delta_i}{\lambda_i} + R_{\text{ext}}; \quad (2)$$

where R_{int} and R_{ext} – is the heat transfer resistance on the inner and external surfaces of enclosure, ($\text{m}^2 \cdot \text{K}/\text{W}$); $\sum R_i$ – is the sum of layers thermal resistance, ($\text{m}^2 \cdot \text{K}/\text{W}$).

The values of R_{int} and R_{ext} determines as follows [8]:

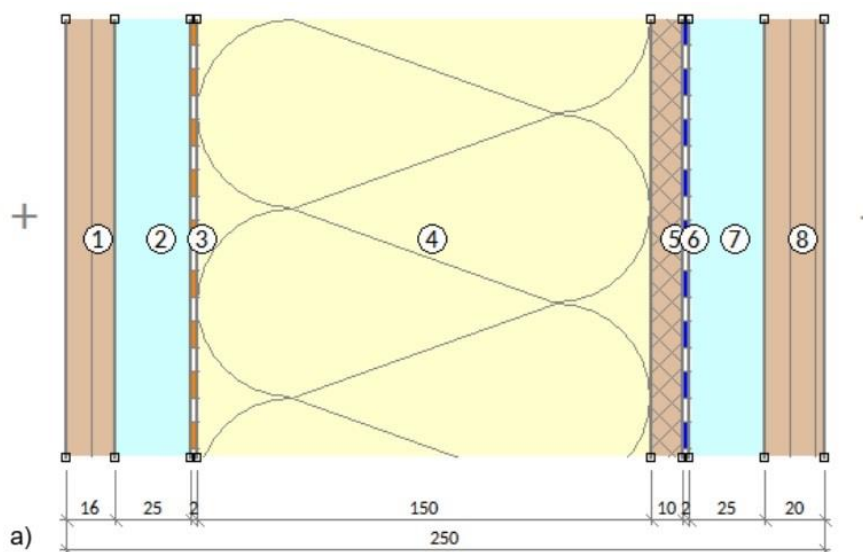
$$R_{\text{int}} = \frac{1}{\alpha_{\text{int}}}; R_{\text{ext}} = \frac{1}{\alpha_{\text{ext}}}; \quad (3)$$

where α_{int} , α_{ext} – the heat exchange coefficients near the internal and external surfaces of enclosure.

According to [6], $\alpha_{\text{int}} = 8.7$ and $\alpha_{\text{ext}} = 23$.

Substituting the values obtained when calculating the (1) and (3) into a formula (2) allows us to obtain the actual values of enclosures heat transfer resistance that given in Table 2.

The standard values of heat transfer resistance for existing types of enclosure structures are determined with accordance to [6].



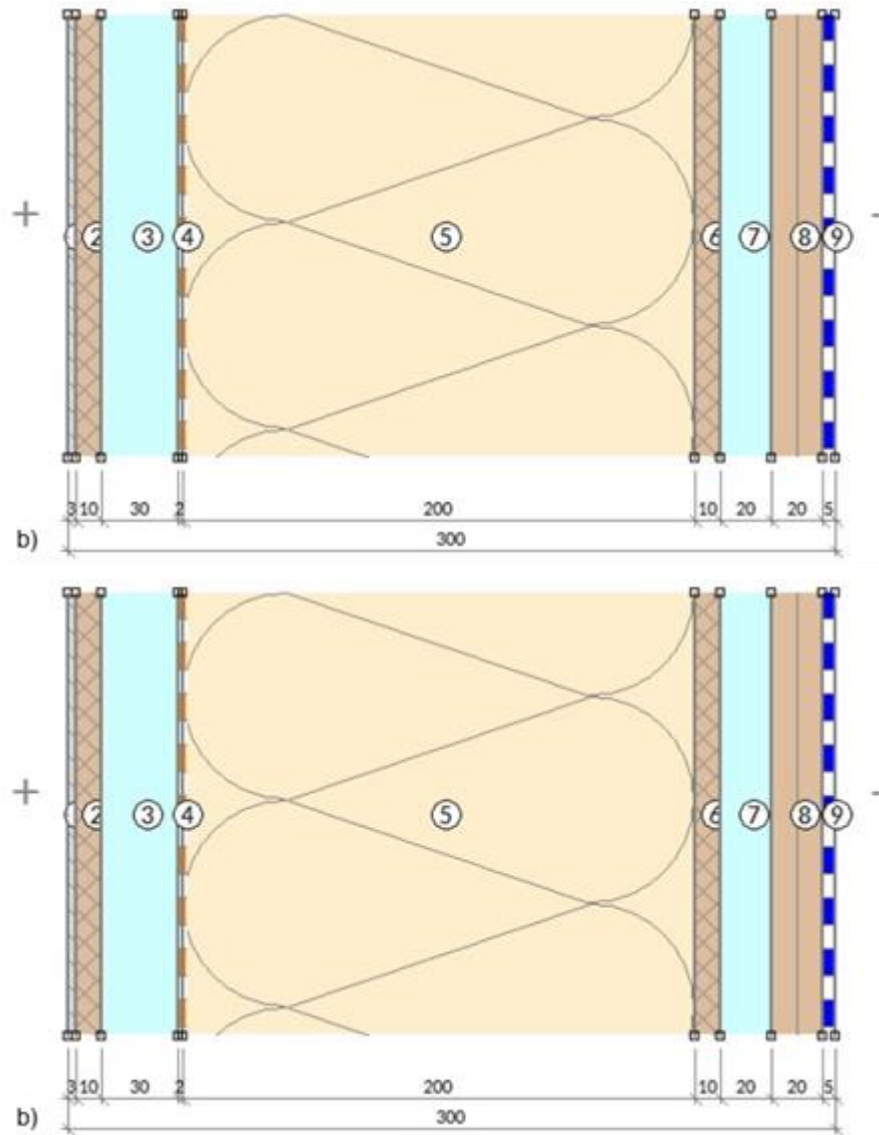


Fig. 7. The calculation models of enclosing structures: a) walls; b) floor; c) roof

Table 2 – Comparison table of standard and actual values of thermal resistance

Type of enclosure	Thermal resistance, $m^2 \cdot K/W$	
	Actual value	Standard value
External walls	4.817	≥ 4
Floor	5.596	≥ 5
Windows	0.48	≥ 0.9
Combined enclosing bordering the outside air	4.76	≥ 7

Analyzing of obtained data, it can be concluded that the 4-10-4-10-4 type windows doesn't meet the regulatory requirements at all. That is also correlates with the results obtained after infrared thermography. In order to improve the energy performances of the windows with accordance to a standard value the 4i-10Ar-4-14Ar-4i type windows with five air filled chambers profile system or better should be installed. According to our calculations such windows system

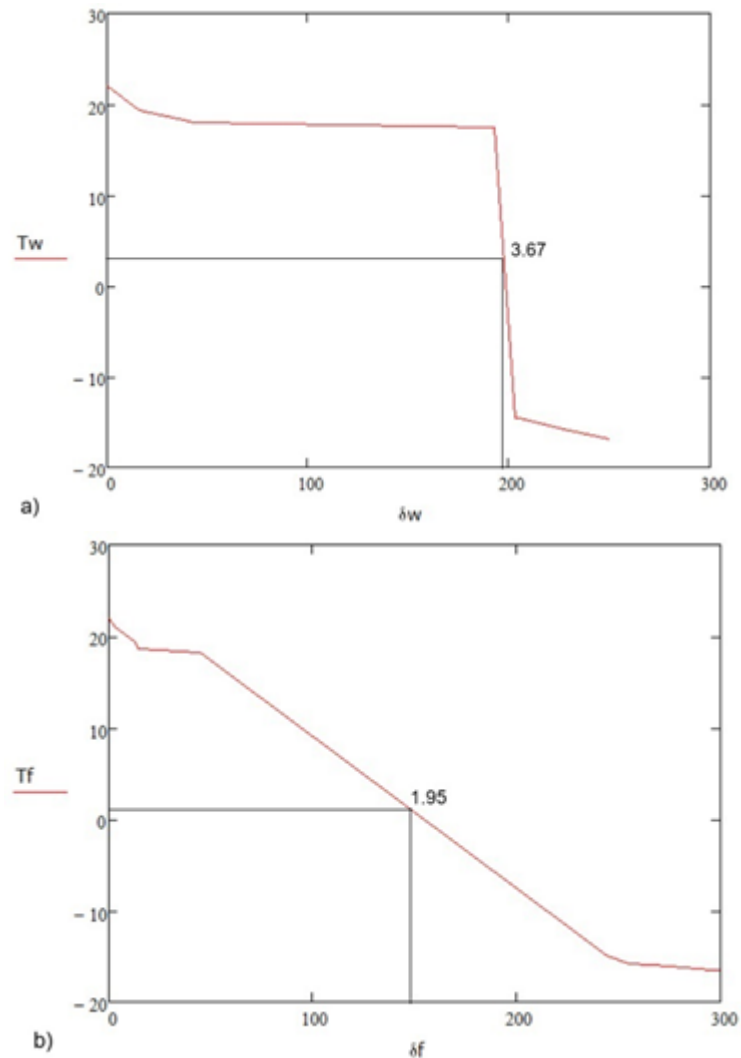
provides the thermal resistance at a level of $0.95 \text{ m}^2 \cdot \text{K}/\text{W}$ with the energy loss about 0.178 kW . Additionally, the thickness of the roofing insulation of 150 mm is not enough. Our calculations have revealed that in order to get the regulatory value the roofing insulation thickness should be increased up to 250 mm .

The temperature distribution through the enclosing layers was calculated as follows [7]:

$$\tau_n = t_{in} - \left(\frac{t_{in} - t_{ext}}{R_0} \right) \cdot \left(\frac{1}{\alpha_{int}} + R_1 + R_2 + R_3 + \dots + R_n \right); \quad (4)$$

where t_{in} – temperature of internal air, $t_{int} = 22 \text{ }^\circ\text{C}$; t_{ext} – estimated temperature of external air, (according to [6], $t_{ext} = -23 \text{ }^\circ\text{C}$); R_0 – overall thermal resistance of enclosure; R_1, R_2, R_3, R_n – thermal resistances of $1^{st}, 2^{nd}, 3^{rd}$ and n layers of enclosure accordingly.

The estimated temperature distribution among the enclosing layers are given in Fig. 8.



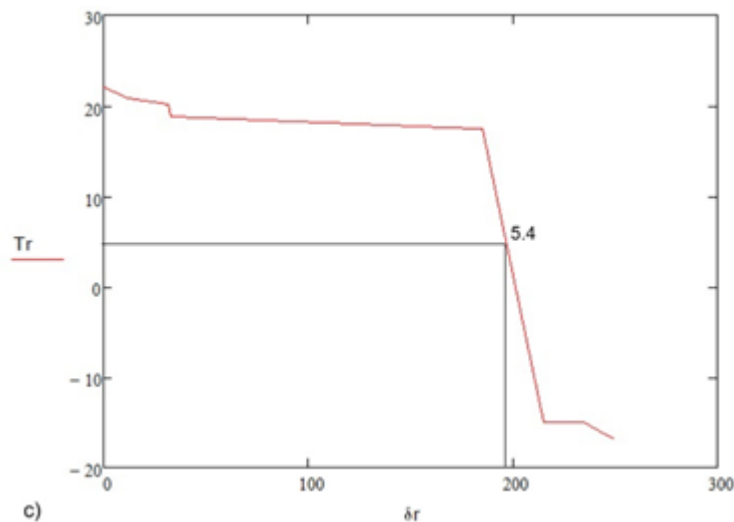


Fig. 8. The calculated temperature distribution among the building's enclosure layers: a) walls; b) floor; c) roof

Analyzing of obtained results, it should be noticed, that the temperature distribution between the walls and the roof layers has similar feature mainly because of similarity in their construction. The average temperatures of the building's enclosures in the period of the coldest month of the year revolve around the positive values, particularly +3.67 °C for walls, +1.95 °C for the floor and +5.4 °C for the roof. Thus, building's envelope provides the positive temperatures inside enclosures without any considerable deviation even though the significant daily and seasonal outside air temperature fluctuations. Theoretically, this should exclude the probability of condensation formation and as a result the possibility of thermal bridges development in the future.

Heat loss through the enclosing structures was determined as follows [7]:

$$Q_{en} = \frac{F}{R_0} (t_{in} - t_{ext}) \cdot n \cdot (1 + \sum \beta); \quad (4)$$

where F – calculated area of enclosing structures, m^2 ; R_0 – thermal resistance of enclosure structure, $(m^2 \cdot K)/W$; t_{in} , t_{ext} – internal and external temperatures, °C; n – the coefficient taking into account the dependence of the position of the outer surface of the fence with respect to the outside air, ($n = 1$) [7]; $\sum \beta$ – the sum of additional heat loss as a share of the main losses, ($\sum \beta = 1$) [7].

External walls area as well as their spatial orientation are given in Table 3.

Table 3 – External walls area

Type	Area by side orientation, m^2							Total
	South	South-West	West	North-West	North	North-East	East	
Walls	31.18	–	12.76	–	34.78	–	12.6	91.32

The heat loss for heating of ventilation air is determined by the formula [8]:

$$Q_{va} = 0.28 \cdot L \cdot \rho \cdot c \cdot (t_{in} - t_{ext}); \quad (5)$$

where L - air flow rate, m³/h (for residential buildings, the specific normalized flow rate is can be taken as 3 m³/h per 1 m² of residential area); ρ - indoor air density, kg/m³; c - specific heat capacity of indoor air, $c = 1.005$ kJ/(kg · °C) [8].

$$\rho = \frac{353}{273 \cdot t_{in}}; \quad (6)$$

The calculation results obtained by calculating of (4) and (5) are shown in the table 4.

Table 4 – Calculation of heat loss through the enclosures and for heating of ventilation air

Type of enclosures	Characteristics of enclosing structures				Heat loss, kW	%, from the overall losses
	Orientation	Dimensions		Area F, m ²		
		a, m	b, m			
Walls	South	8.62	3.7	31.9	0.582	13
	West	3.45	3.7	12.76	0.238	5
	North	9.4	3.7	34.78	0.649	14.6
	East	3.45	3.65	12.6	0.235	5.3
Total:		24.92	14.75	91.3 2	1.3	31.9
Floor	–	8.99	3.04	27.3	0.404	9.9
Roof	–	8.99	3.04	27.3	0.515	12.6
Windows	Left	1.2	1.2	1.44	0.343	7.245
	Right	1.2	1.2	1.44	0.343	7.245
Total:		1.44	1.44	2.88	0.686	16.8
Total loss through enclosures:				148.8	2.9	71.3
Heat loss for heating of ventilation air:					1.165	26.3
Overall heat loss:					4.43	

5. Conclusions

It can be concluded that the walls are the main source of the heat loss with the overall balance of 31.9 % from the building`s total heat loss. Increasing the thickness of the wall insulation up to 200 mm leads to a reduction in the percentage of heat loss to 27% of the total amount.

Heat loss for heating of ventilation air is the second source of the building`s heat loss that is 28.6 % from the total heat loss.

With the existing characteristics of translucent structures with the thermal resistance within 0.48 – 0.52 ((m²·K)/W), heat loss through the windows is 16.8 %. It should be noted that the 4-10-4-10-4 type windows doesn`t meet the regulatory requirements and must be replaced with more

energy efficient ones. Therefore, to meet regulatory requirements for thermal insulation properties, windows must have at least a 4i-10Ar-4-10Ar-4i design, i.e. they must have a soft selective coating on the outer and inner glass, and the air gaps must be filled with an inert gas, particularly argon. Bringing the thermal resistance of windows to the standard values of 0.95 ((m²·K)/W) reduces the heat loss through the windows up to 4.39 %.

The floor and ceiling have almost the same heat loss ratios of 9.9% and 12.6%, respectively.

References

1. Bolotov, M., Bolotov, H., Prybytko, I., Korzachenko, M. (2024). Sustainable practices of concrete manufacturing. *Technical sciences and technologies*, (2(36)), 15-30. [DOI: 10.25140/2411-5363-2024-2\(36\)-15-30](https://doi.org/10.25140/2411-5363-2024-2(36)-15-30)
2. Blismas, N., C. Pasquire, and A. Gibb, Benefit evaluation for off-site production in construction. *Construction Management and Economics*, 2006. 24(2): p. 121-130.
3. QHome. QHome-26. URL: <https://qhome.ua/project/qhome-26/> (дата звернення 07.03.2025)
4. Designing Buildings. Thermal conductivity. URL: https://www.designingbuildings.co.uk/wiki/Thermal_conductivity (дата звернення 28.04.2025)
5. International Organization for Standardization. Building Components and Building Elements – Thermal Resistance and Thermal – Calculation Methods, International Organization for Standardization; ISO 6946; International Organization for Standardization: Geneva, Switzerland, 2017
6. DBN V.2.6-31:2021. Thermal insulation and energy efficiency of buildings. Kyiv. 2022
7. Hayduk, O.V., Herlyand, T.M., Kulalayeva, N.V., Pivtoratska, N.V., Pyatnychuk, T.V. (2021). Technologies of insulation of building facades: a textbook. Zhytomyr: Polissya, 362. [doi: 10.32835/978-617-8117-00-9/2021](https://doi.org/10.32835/978-617-8117-00-9/2021)
8. Paraschiv, L., Paraschiv, S., Ion, V. (2017). Increasing the energy efficiency of buildings by thermal insulation. *Energy Procedia*. [doi:10.1016/j.egypro](https://doi.org/10.1016/j.egypro).

3. OPEN BIM-BASED CO-SIMULATION FOR BUILDING ENERGY AND LCA: TWO BUILDINGS IN THE UK AND LUXEMBOURG

Tytarenko R. Yu., Chmil R.Ye., Pozniak O. R.

1. ABSTRACT

Given the complexity and interconnection of factors in building evaluation, a key challenge is the integration of multiple domain models (e.g., thermal comfort, indoor environmental quality, and occupant comfort) to better inform strategies for improving building performance. Only one previous work attempted to combine building energy simulation with life cycle assessment; none has done so dynamically. A dynamic, open BIM-based co-simulation architecture enables the tight integration of EnergyPlus and Brightway2 without relying on heuristics or simplified tools. It provides time-differentiated results and is built entirely on open technologies. The architecture was validated on two non-domestic buildings in the UK and Luxembourg, demonstrating applicability to construction and operational life cycle phases. Dynamic co-simulation across energy and life cycle domains allows a more holistic and detailed building performance analysis.

2. INTRODUCTION

Given the complex interdependencies affecting building performance, integrating multiple domain models (e.g., thermal comfort, indoor environmental quality, occupant satisfaction, acoustics, visual comfort, glare prevention, aesthetics) is essential for a holistic evaluation. Co-simulation, which combines coupled simulations into a unified model, provides a promising approach. While life cycle assessment (LCA) is increasingly used to evaluate high-performance buildings, most prior work treats building energy simulation (BES) and LCA separately, offering no integrated perspective.

An open BIM-based co-simulation architecture enables the tight integration of BES and LCA, supporting dynamic, time-differentiated results. Two non-domestic buildings in the UK and Luxembourg were simulated across construction and operational phases, exploring three scenarios each. This approach enhances understanding of building performance, addresses interoperability challenges between BIM, BES, and LCA tools, and provides a repeatable methodology for practitioners. It streamlines co-simulation, facilitates scenario analysis, and supports the achievement of environmental targets, with applicability for designers, managers, researchers, and policymakers worldwide.

3. BACKGROUND

The following provides background on BES, BIM interoperability, and LCA. BES supports designers and ensures building compliance, but these simulations are typically conducted outside the BIM framework. The integration of BES with BIM involves numerous interoperability challenges. In the building sector, LCA is used to evaluate environmental impacts and resource consumption. These concepts are well-established and are not included in the systematic review in the following section [15].

3.1. Building Energy Simulation (BES)

Building energy simulation (BES), building energy modelling (BEM), thermal modelling, and building performance simulation all involve creating a digital model of a building and simulating it in energy software. Despite BES being used since the 1960s, it remains poorly integrated into early design stages, often limiting energy-efficient strategies. Manual input of building data is time-consuming, leading to the development of BIM-to-BEM approaches to improve efficiency and data flow.

Geometry creation is the most time-intensive task, followed by assigning constructions, internal gains, and schedules. BIM-to-BEM tools vary in automation and data integrity, but no approach fully achieves automatic, lossless conversion. Semi-automated tools, such as those by O'Donnell et al. [9], reduce time and cost, allow rapid design alternatives, improve simulation accuracy, and produce models with lower energy consumption. These benefits can be grouped as efficiency gains and technical improvements.

However, interoperability remains a significant challenge. Studies by Alsharif [1] and Elagiry [5] identify two main issues: (1) IFC geometry generation, which often requires manual correction, and (2) data enrichment, where essential information on site, geometry, materials, systems, internal gains, and building operation is incomplete, unavailable, or not transferable. Effective BIM-based BEM automation is considered crucial for advancing design and construction practices.

3.2. Building Information Modelling (BIM)

Building Information Modelling (BIM) represents buildings as 3D graphical models containing information on geometry, properties, names, and component functions. The construction industry is moving from traditional 2D drawings toward BIM. BIM is widely used as a basis for “BIM-to-BEM” approaches and for LCA, with IFC often converted to gbXML for semi-automated workflows.

Several approved energy simulation tools exist, including TAS, ApacheSim, EnergyPlus (E+), and SBEM. SBEM is a simplified tool, while the others are dynamic simulation models. Only E+ is free and open-source, making it the preferred tool. E+ uses the .idf file format, with OpenStudio Application (OSA) as its standard GUI frontend, which uses .osm files converted to .idf at simulation.

3.3. Environmental Life Cycle Assessment (LCA)

Environmental Life Cycle Assessment (LCA) is a widely used methodology for evaluating the ecological impacts and resource use of products, services, and processes across their entire lifecycle, guided by ISO 14 000 standards. A key aspect is the functional unit, which allows meaningful comparison between alternatives providing the same function. LCA considers the entire lifecycle, avoiding burden shifting and supporting trade-off analysis.

LCA is well established in the building sector, applied from single buildings to entire districts. It can inform design decisions during early stages and guide operational strategies. Standard professional LCA tools include SimaPro, openLCA, GaBi, and Umberto, with advanced programming possible via Brightway2 (BW2). High-quality background data, typically from databases like ecoinvent, is essential.

Future research highlights the potential of BIM and digital twins to provide accurate, detailed data for LCA. Combined with co-simulation and sensor data, this enables precise modeling of building-occupant interactions and integration of results into the LCA framework.

4. CURRENT ADVANCES IN CO-SIMULATION FOR THE BUILDING SECTOR

The study by Yeung et al. [15] presents the concept of co-simulation. It provides a systematic review of the state of the art in the building domain, highlighting existing approaches and their contributions. The review follows the PRISMA protocol [10], covering publications from March 2017 to January 2023, with journal and review articles retrieved from Semantic Scholar using the search string: (ALL=(co-simulation)) AND ALL=(buildings).

The initial search returned 392 records. After removing non-journal/review articles, duplicates, and applying screening criteria focused on studies coupling different building domain models to describe subsystem interactions, 21 articles were included in the final review. Five exclusion criteria were used:

- 1) lack of cross-domain coupling;
- 2) no co-simulation approach;
- 3) no advancement of the state of the art;
- 4) duplicates or highly similar works;
- 5) studies only comparing previously developed approaches with traditional methods.

Systematic review methodology (according to [15], based on the PRISMA 2020 statement flow diagram) is presented in Fig. 4.1.

Co-simulation is applied in multiple fields such as automotive, energy, HVAC, IC, maritime, and robotics [6], enabling integration of independently developed partial solutions that are otherwise difficult to combine with monolithic tools [13]. It allows simultaneous simulation of subsystems through separate simulators acting as black boxes. Advantages include tool flexibility, rapid software testing, distributed teamwork, and multi-scale analysis [14].

In building simulation, co-simulation links sub-models (thermal, airflow, daylighting, etc.) to capture interactions during runtime [13]. Coupling strategies can be sequential or bi-directional, with loose or strong feedback, while techniques include one-to-one links, middleware (e.g., BCVTB, RabbitMQ), or standardized interfaces like FMI [2; 8]. Internal vs. external coupling is another distinction (e.g., CFD within BES vs. data exchange between existing tools).

Building benefits include modelling occupant behaviour, evaluating subsystem trade-offs, and real-time envelope simulations using E+ or TRNSYS. Recent approaches also integrate machine learning for predictive control [7; 12; 16]. However, challenges remain: a lack of standardization and interoperability between BES tools, and the need for advanced programming and domain knowledge [13].

Some BES tools support BIM integration via IFC, while parametric platforms like Grasshopper and Ladybug Tools enable loosely coupled co-simulations and performance-driven design [3]. These workflows can further link results to costs and GHG emissions across the building lifecycle [11].

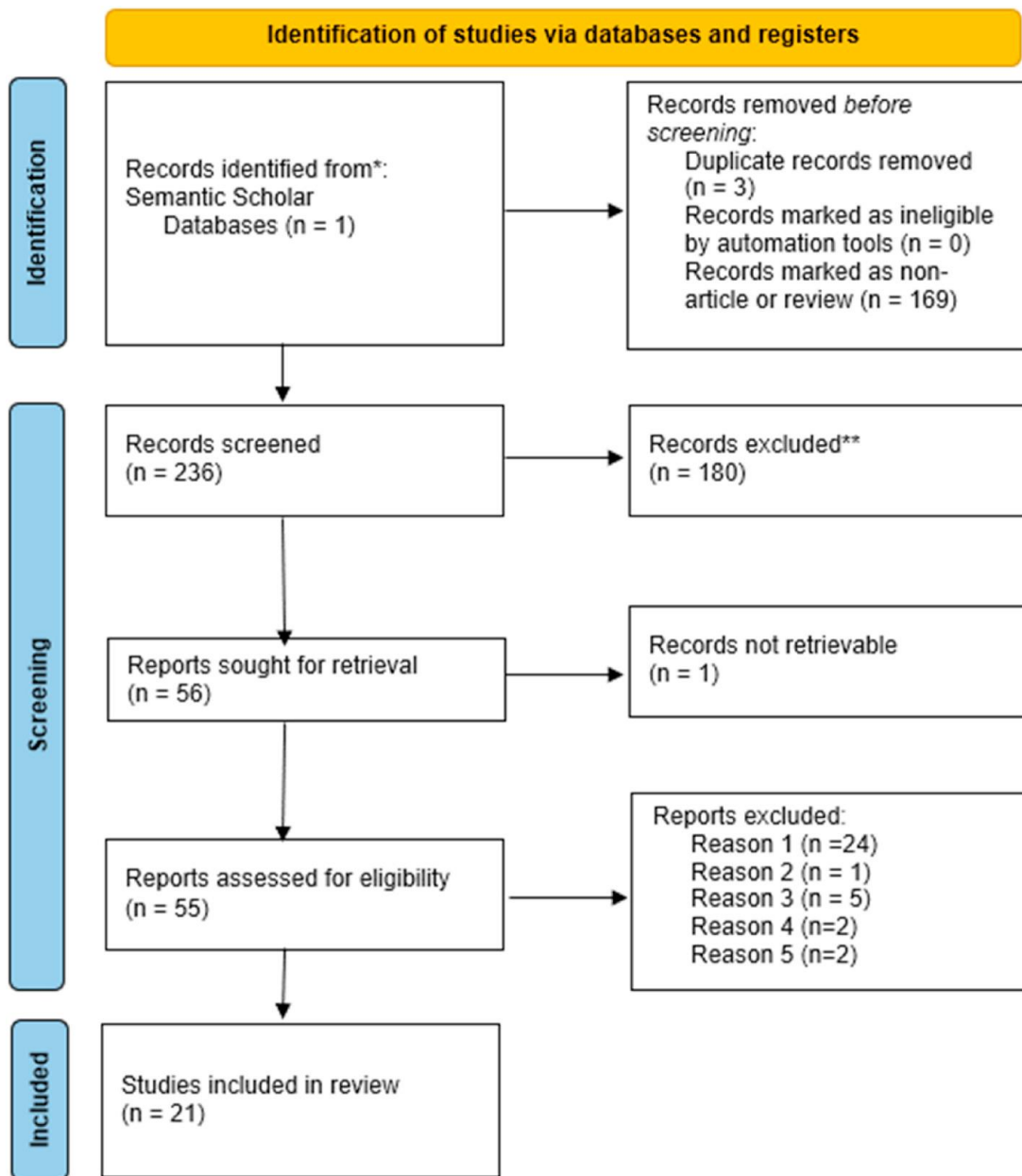


Fig. 4.1. Systematic Review Methodology
(According to [15], Based on the PRISMA 2020 Statement Flow Diagram)

A brief overview of studies grouped by domain-to-domain coupling is presented in Table 4.1.

Table 4.1.

Energy and Climate	Since 2017, research has combined building energy models with indoor climate, urban climate, and ventilation domains using TRNSYS, COMSOL, E+, ENVI-met, FLUENT, and CONTAM. Applications include wall heat transfer, green roofs and facades, ventilation control, retrofit analysis, and daylighting. Results demonstrated benefits in energy reduction, improved thermal conditions, and health outcomes.
Energy and Occupant Behaviour/Health	Agent-based models coupled with E+ and Radiance assessed occupant influence on energy use. Several studies addressed COVID-19, showing that optimal space layouts and ventilation strategies reduced infection risk and energy consumption.
Energy and Acoustics	A 2021 study integrated sound insulation, energy efficiency, and cost optimisation for nearly zero-energy homes.
Energy and Life Cycle	Cellura et al. [4] developed an LCA tool linked to TRNSYS, which was validated with a residential case study in Italy, noting the lack of full BES–LCA integration.
Occupant Behaviour and Fire Safety	A 2021 BIM-based co-simulation combined fire dynamics with agent-based occupant modelling, applied to a school building in Montreal to evaluate renovation plans considering evacuation times, costs, and schedules.

This review highlights that while integrating different domain models to capture interactions among building elements is crucial for advancing building research, a lack of studies thoroughly combine BES and LCA, particularly those with dynamic capabilities. Most existing works treat these approaches as separate methodologies, and only one study has attempted their integration, but none have done so dynamically [4]. In turn, the analyzed study [15] introduces an open BIM-based co-simulation architecture with time-differentiated capabilities to address this gap. This is the first architecture to integrate E+ and Brightway2 tightly without relying on heuristics or simplified tools. Moreover, it is the first BES–LCA co-simulation framework to provide dynamic LCA results and implement it entirely with open-source technologies (E+ and BW2).

5. CO-SIMULATION FRAMEWORK

The following describes each component of the open BIM-based co-simulation architecture developed in the study by Yeung et al. [15] Fig. 4.2 presents a use case diagram showing how designers and managers can utilize the architecture, while Fig. 4.3 depicts its overall structure. In the following sections, the arrows in Fig. 4.3 are called processes.

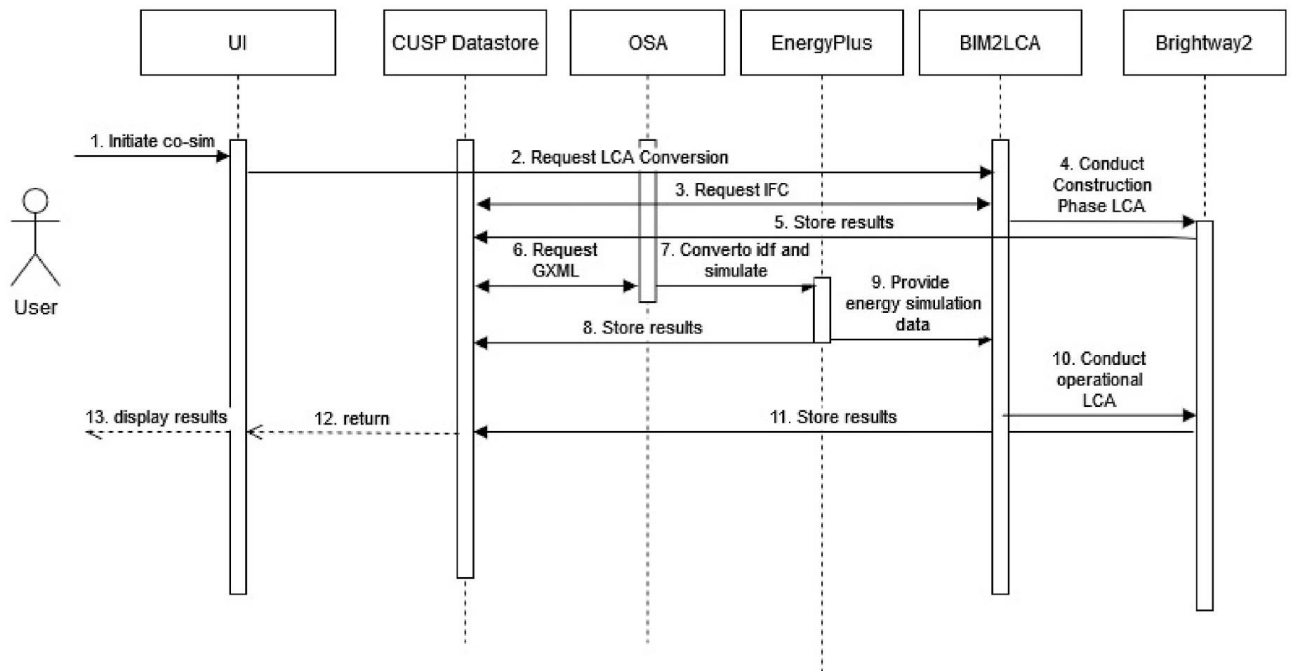


Fig. 4.2. Use Case Diagram for Co-Simulation

5.1. Component 0: BIM Authoring Tools

Component 0 (see Fig. 4.3) covers conversion to IFC and BIM enrichment (process 0.1). Although a single process, it is shown as two steps to highlight compatibility with multiple BIM authoring tools (e.g., Revit, ArchiCAD, or any software capable of IFC export). A prerequisite is the availability of an IFC building model.

The co-simulation architecture sets specific BIM data requirements. Since BIMs are usually created for purposes other than BES, additional detail is needed:

- All rooms must be defined as “spaces” with assigned types (corridor, office, plant room, WC, etc.).
- Thermal zones must be specified for individual rooms or groups.
- Materials and constructions must be defined.

Steps 1–2 are common in well-developed BIMs, while step 3 often requires further enrichment. Even if IFC exports don’t perfectly capture geometry, defining spaces resolves this. The authors treat these requirements as prerequisites, since BIMs typically already include spaces and element properties before BES modeling, avoiding unnecessary duplication later.

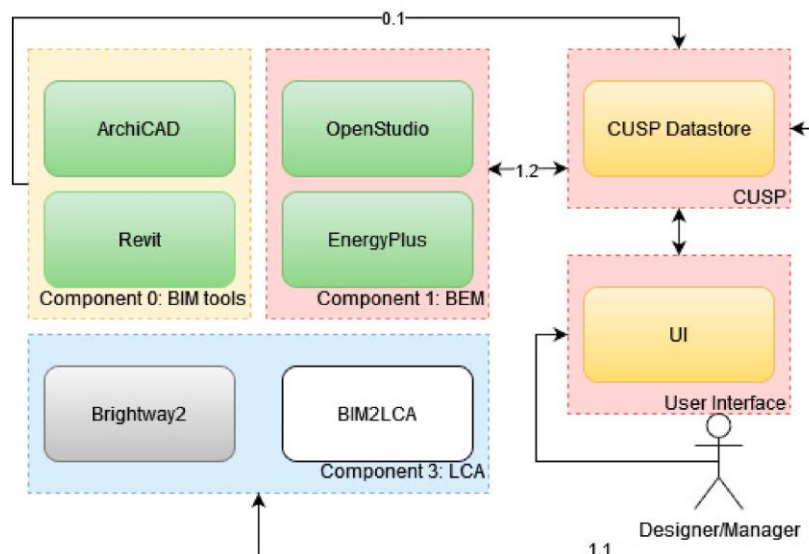


Fig. 4.3. Open BIM-Driven Co-Simulation Framework

5.2. Component 1: Interoperability Processing

BIM to BEM. The enriched BIM model in IFC format is then exported into a model schema that can be imported into an energy simulation tool. An automated translation tool was developed to convert an IFC file into either a Green Building XML (gbXML) or HBjson file (process 1.2).

BEM Enrichment. The gbXML or HBjson file is then imported (process 1.2) into the user interface. First, a weather file is retrieved from an online service and linked to the imported gbXML/HBjson model. At this stage, BEM enrichment is performed through the user interface shown in Fig. 4.4, where building service systems and internal loads are input.

5.3. Component 3: Life Cycle Assessment (LCA)

The enriched BIM in IFC format is also used for LCA calculations (process 1.1). The BIM2LCA tool parses the IFC file to extract material data for building constructions and maintains links between activities (e.g., concrete mixing) in a BW2 project and material names. Materials are retrieved from ecoinvent 3.8, and BIM2LCA includes default methods from the Environmental Footprint 3.0, an established framework maintained by the European Joint Research Centre since 2007 (summarized in Table 4.2). This ensures a standard set of impact categories and indicators, though users can customize methods and associations.

BW2 performs the LCA calculations for outputs from the BES (process 2.2). For example, CTU quantifies human toxicity impacts, and water use is expressed in “World eq. deprived” via the AWARE method, accounting for water available after human and ecosystem needs.

Users interact with the co-simulation through the interface, which provides access to simulation results, allows modifications, and enables running new simulations in E+ or executing LCA via BW2.

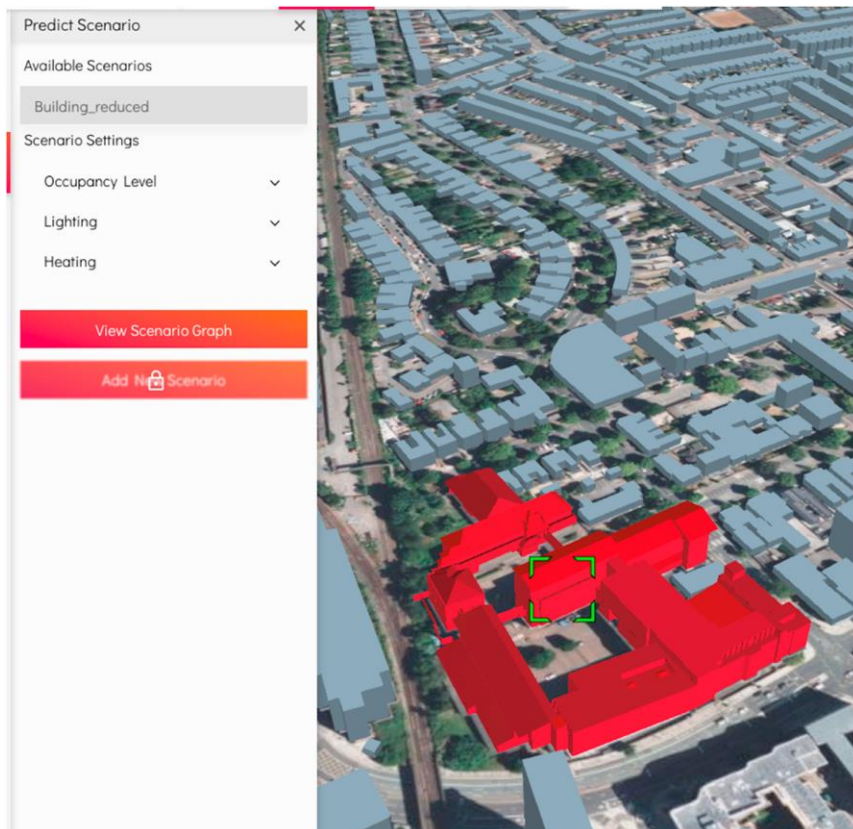


Fig. 4.4. Interface for User Interaction

5.4. Co-simulation Execution and Output

The simulation generates an .idf file sent to the CUSP semantic data store (process 1.2) along with energy mix data. BIM2LCA then performs the operational LCA and stores the co-simulation results in CUSP. These results for both construction and operational phases are displayed to the user.

By integrating whole-building BES and LCA, the co-simulation provides time-differentiated energy data, allowing dynamic representation of the building's operational phase. The following sections present case-study validations illustrating these time-differentiated results.

Table 4.2.

Impact Category	Indicator	Unit
Water use	User deprivation potential	m ³ world eq. deprived
Photochemical oxidant formation: human health	Tropospheric ozone concentration increase	kg NMVOC eq
Particulate matter formation	Human health effects associated with exposure to PM2.5	Disease incidences
Material resources: metals/minerals	Abiotic resource depletion	kg Sb eq
Land use	Soil quality index	Dimensionless
Human toxicity: non-carcinogenic	Comparative Toxic Unit for humans (CTUh)	CTUh
Human toxicity: carcinogenic	Comparative Toxic Unit for humans (CTUh)	CTUh
Eutrophication: terrestrial	Accumulated Exceedance (AE)	mol N eq
Eutrophication: freshwater	Accumulated Exceedance (AE)	kg P eq
Energy resources: Non-renewable	Abiotic resource depletion – fossil fuels (ADP-fossil)	MJ
Climate change	Radiative forcing as Global Warming Potential (GWP100)	kg CO ₂ eq

6. CASE-STUDY VALIDATIONS

The co-simulation framework from above [15] was validated using two buildings: a university building in Cardiff, UK (3 floors, 3 728 m²) and an office building in Luxembourg (5 floors, 9 231 m²). The university model was created in Revit, the office in ArchiCad, confirming that the workflow works with both formats.

Figs. 6.1 and 6.2 show the IFC models and resulting spaces imported into OSA.

Three annual scenarios were simulated for each building: 1) Building_Base – typical occupancy and operation; 2) Building_reduced – occupancy and operation reflecting 2020, affected by COVID-19; 3) Building_reduced 0 ht – as scenario 2, but with reduced heating in unoccupied spaces.

These scenarios analyzed the benefits of time-differentiated LCA versus static results, highlighting the effect of operational variations on environmental impacts. Electricity and heating consumption were simulated via the CUSP interface with BW2, integrating LCA results into the BES and feeding changes back into the BIM, providing a more accurate representation of operational environmental impact.

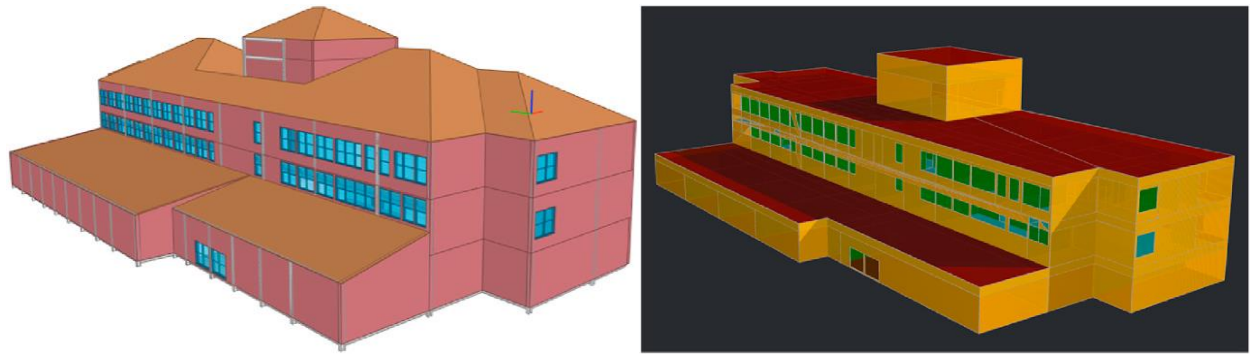


Fig. 6.1. University Building IFC Model (Left) and Imported Spaces in OSA (Right)

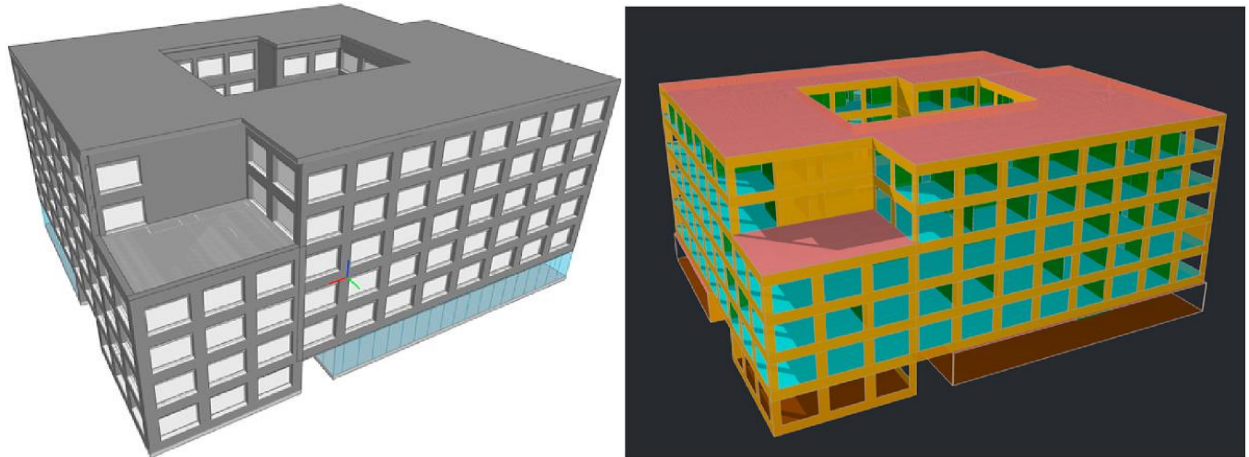


Fig. 6.2. Office Building IFC Model (Left) and Imported Spaces in OSA (Right)

6.1. Case Study 1: University Building, Cardiff

Fig. 6.3 presents the LCA results for the university building's material footprint and energy consumption, assuming a 50-year lifespan. Material demand dominates most impact categories, including land use, water use, and effects on human health and ecosystems, highlighting the importance of construction materials. Heating and electricity consumption, however, are significant for energy resources and climate change, illustrating the trade-off between materials and operational energy.

Fig. 6.4 breaks down environmental impacts by material type. Concrete contributes most (43–92 %) due to its high footprint and the BIM process prioritizing structural elements. Gypsum dominates the impact of material resources (50 %), while steel contributes 14–36% to the carcinogenic effects.

Fig. 6.5 shows climate change impacts under three scenarios: Base, Reduced, and Reduced_0 ht. Reduced occupancy and operation cut total environmental impact by 56 %, rising to 61 % with lower heating loads. Due to the linear LCA framework, similar reductions occur across categories. GWP plots in Figs. 6.5 and 6.9 are simplified to monthly data points; complete 10-minute timestep data are accessible via the user interface.

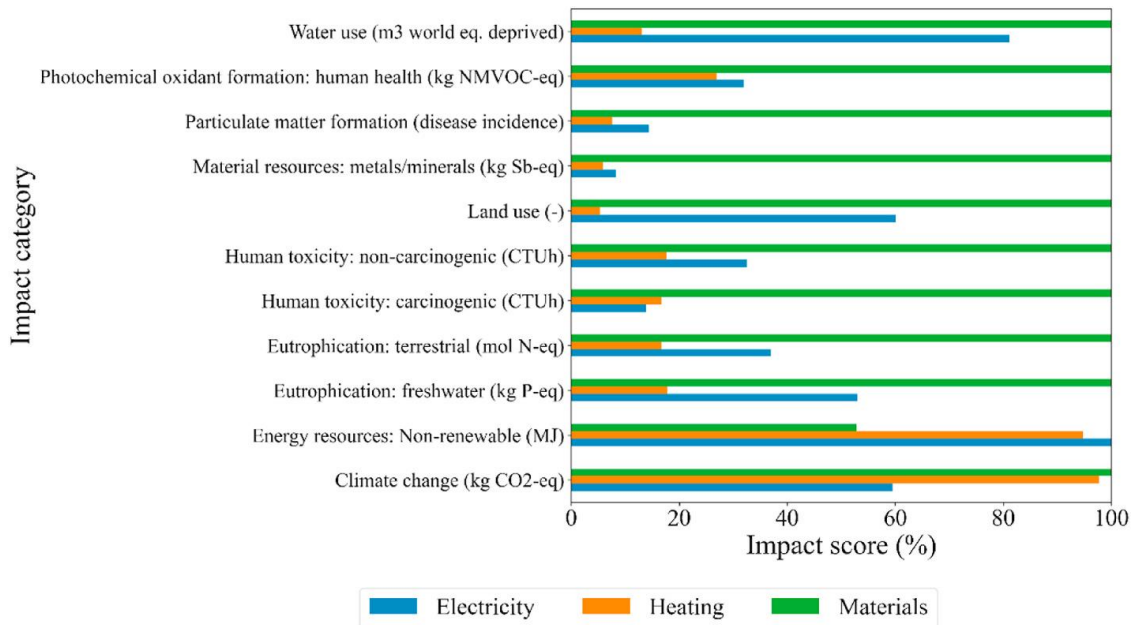


Fig. 6.3. Life Cycle Impact Assessment Results for the University Building, Showing the Contributions of Electricity, Heating, and Materials Over the Evaluated Period (Results Are Normalized Relative to the Highest Value in Each Impact Category)

Comparing Figs. 6.3–6.4 (static LCA) with Figs. 6.5–6.6 (time-differentiated LCA) highlights the latter’s benefit: designers can see which elements have the highest impact and when they do, allowing targeted interventions. For example, operational changes had a high GWP impact in May but a minimal effect in December.

The framework also allows scenario-based adjustments to material demand through the UI (see Fig. 4.4), though this was outside the scope for an already constructed building. Such functionality would be most relevant during the design phase of new projects.

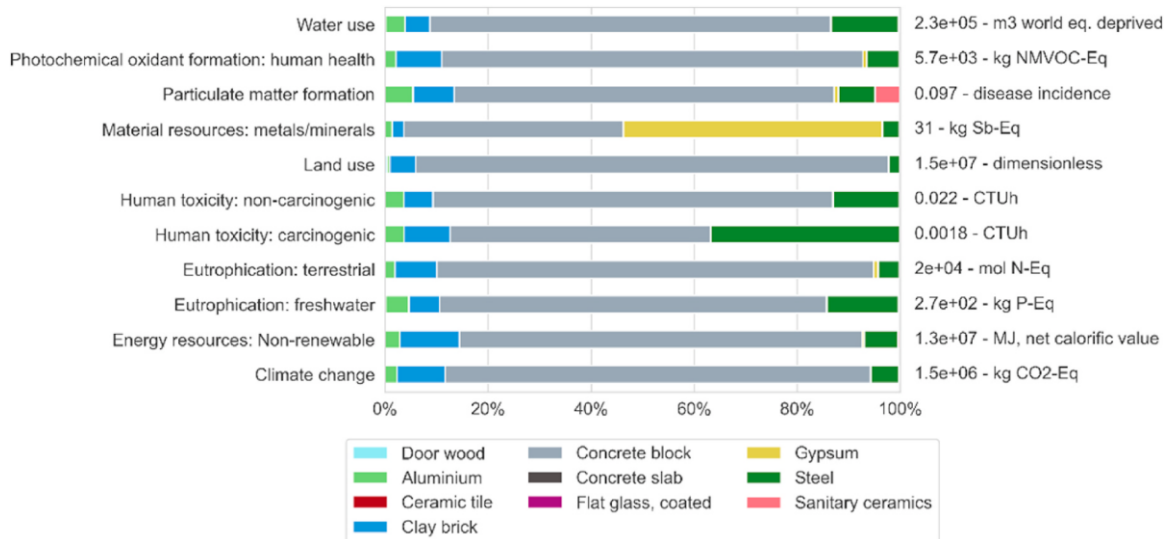


Fig. 6.4. Analysis of How Different Material Types Contribute to the Overall Environmental Impact of Construction Material Demand

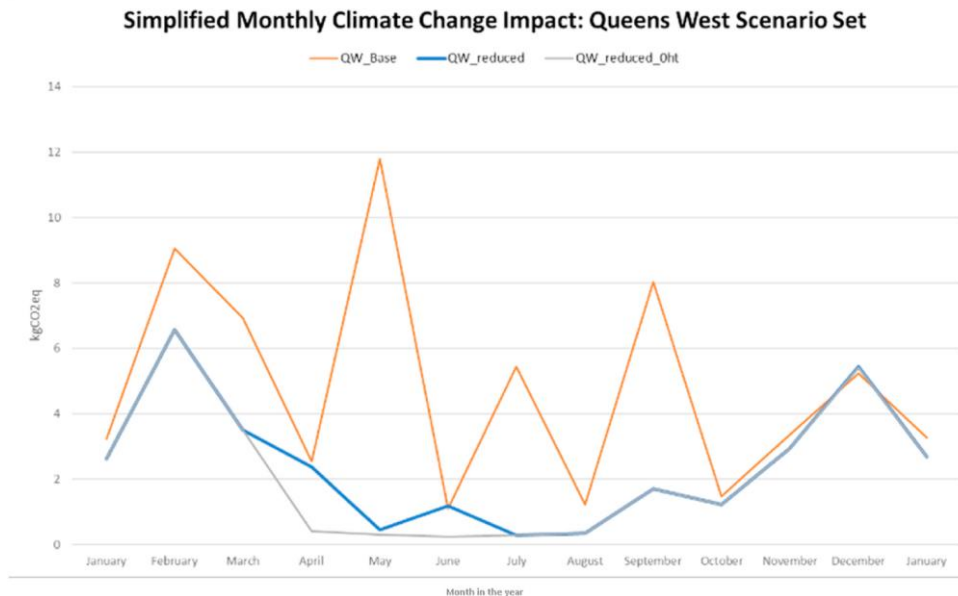


Fig. 6.5. Annual Time Series of LCA Results for the University Building’s Global Warming Potential (kg CO₂eq) Across the Three Assessed Scenarios

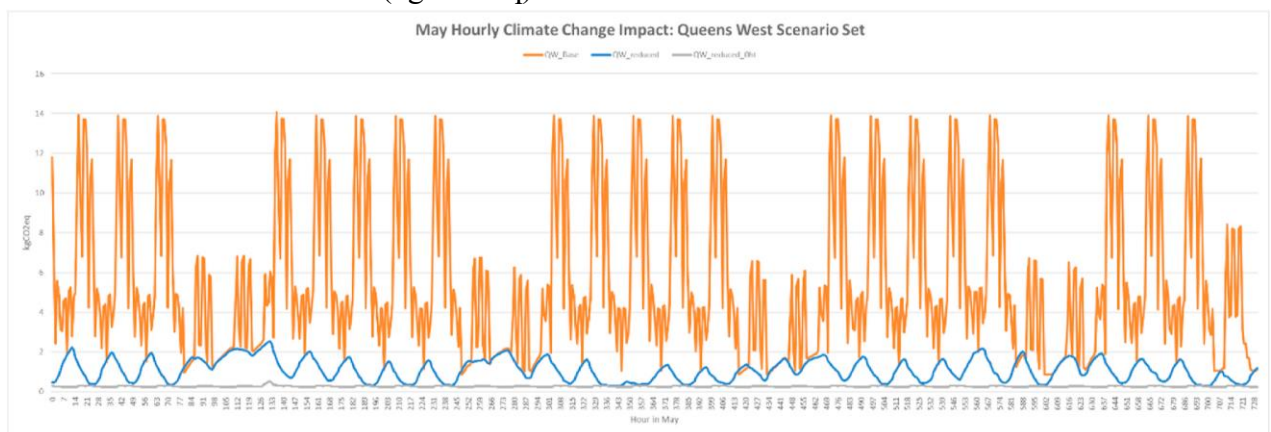


Fig. 6.6. Hourly Climate Change Impact for the University Building in Cardiff During May

6.2. Case Study 2: Office Building, Belval

Fig. 6.7 shows the LCA results for the building’s materials and energy use. As in the previous case, materials dominate most impact categories, while energy use is more relevant for climate change and energy resources. Higher electricity and heating consumption from the BES increases their contribution to most categories, including climate change and water use. Including additional materials in the BIM, such as plywood (see Fig. 6.8), also raises the materials’ impact in categories like land use. Unlike the Cardiff case, steel is absent, highlighting the dependence of results on BIM quality.

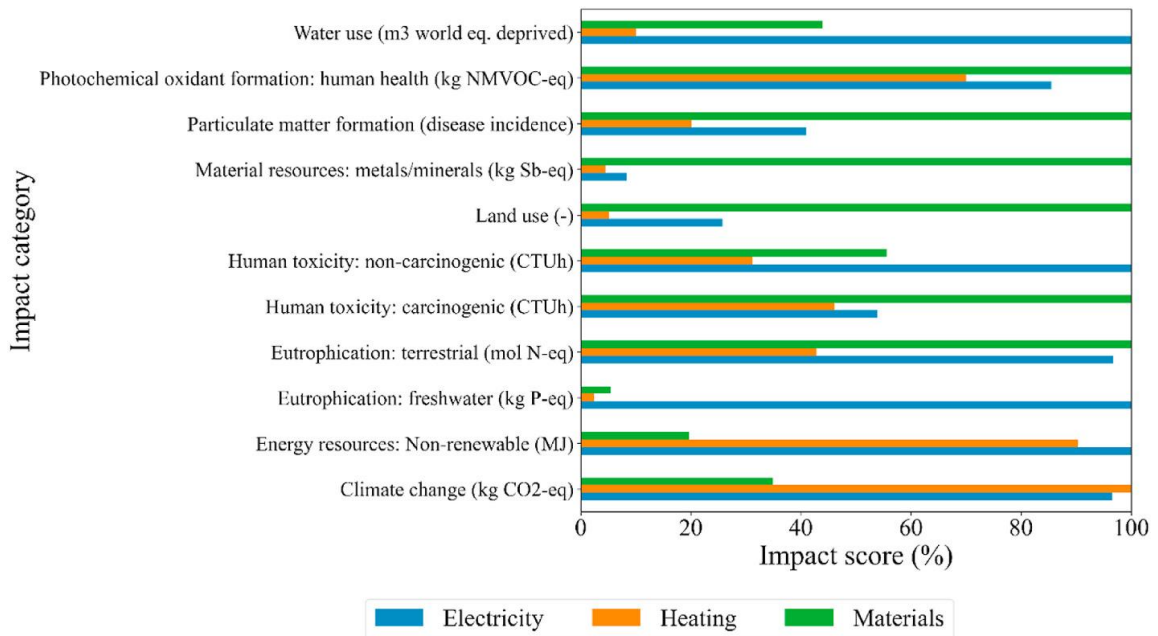


Fig. 6.7. Life Cycle Impact Assessment Results for the Office Building, Showing the Contributions of Electricity, Heating, and Materials Over the Evaluated Period (Results Are Normalized Relative to the Highest Value in Each Impact Category)

Fig. 6.9 presents the impacts of climate change under three scenarios. Reducing occupancy and operation in the “Reduced” scenario lowered the total effect by 72 %, increasing to 74 % in “Reduced_0 ht” due to lower heating demand. Time-differentiated results (see Figs. 6.9–6.10) show interventions strongly affect February–May, less so in June and November, with some daily interventions having minimal effect.

Time-differentiated LCA allows for more precise assessment of environmental impacts over time, showing when benefits occur and enabling designers and managers to better target improvements.

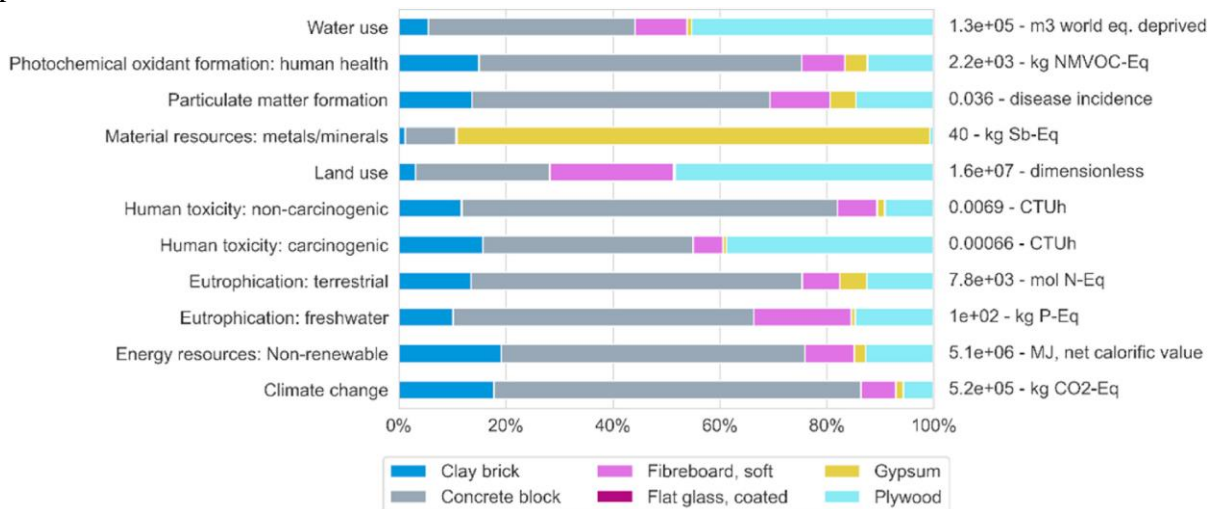


Fig. 6.8. Analysis of How Different Material Types Contribute to the Overall Environmental Impact of Construction Material Demand

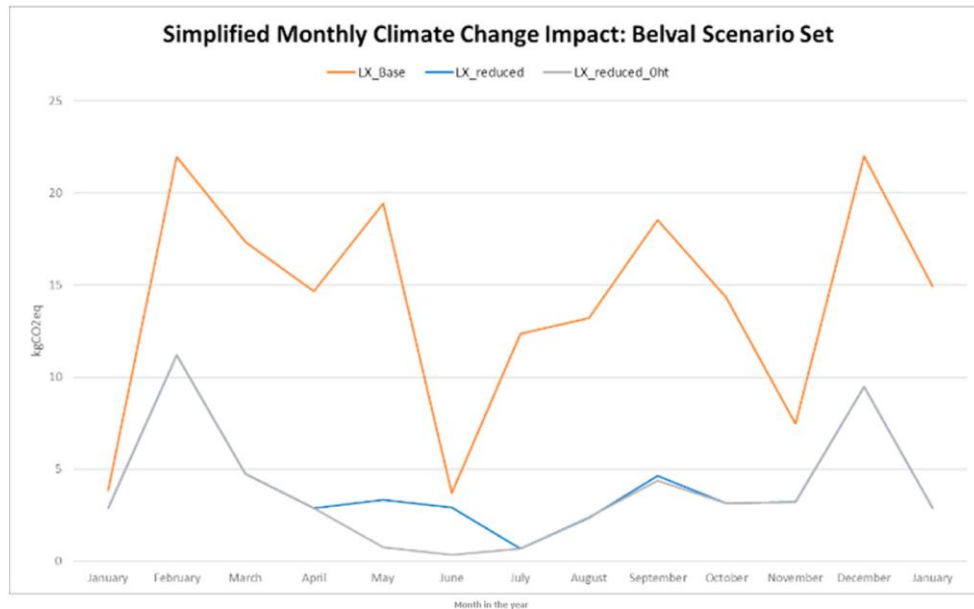


Fig. 6.9. Annual Time Series of LCA Results for the Office Building’s Global Warming Potential (kg CO₂eq) Across the Three Assessed Scenarios

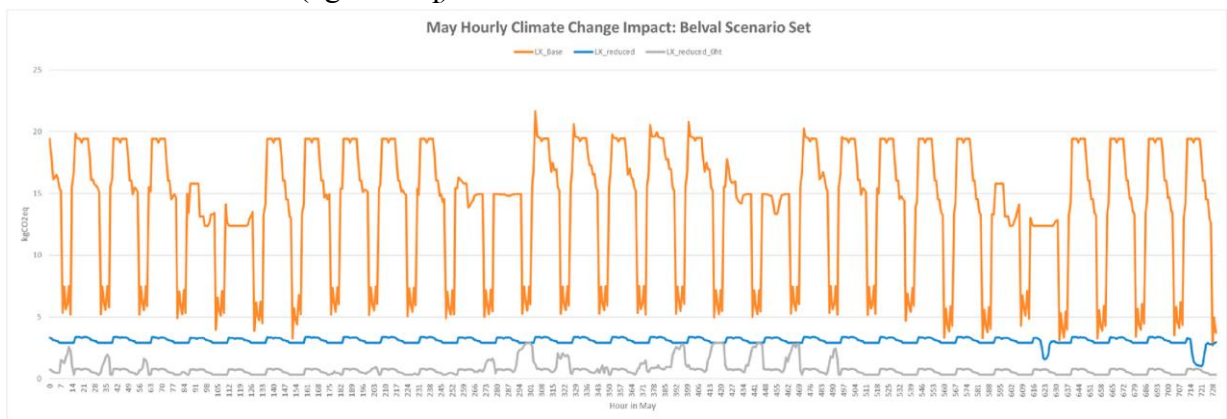


Fig. 6.10. Hourly Climate Change Impact for the Office Building in Belval During May

7. FINDINGS

Yeung et al. [15] developed and validated an open BIM-based co-simulation architecture integrating BES (E+) and LCA (BW2) for entire buildings across construction and operation phases. It is the first approach to tightly integrate these tools, including a nationally approved UK BES software, enabling a dynamic inventory for electricity, heating, and water. This allows time-differentiated, more accurate results and granular analysis at specific points in time.

Applying the co-simulation from the BIM creation stage reduces labour, time, and costs by avoiding geometry recreation in multiple software tools. Its open BIM nature lowers capital barriers, making BES-LCA co-simulation accessible to individuals and smaller organisations, and encourages BIM adoption. Geometry creation, internal gains, and schedules, often the most time-consuming tasks, are streamlined through automated conversion and the integrated UI.

The study tested three scenarios representing building use during COVID-19, confirming benefits like rapid testing, parallel sub-system modelling, and scalability. Overall, the methodology provides a repeatable, practical framework for dynamic LCA from BIM, reducing time and cost while enabling scenario analysis and supporting environmental targets globally for researchers, designers, managers, and policymakers.

8. LIMITATIONS

The authors of the above study examined the limitations of the developed co-simulation architecture. Although an “open” approach was implemented, it was applied to BIM models created with proprietary tools. Geometric inconsistencies were addressed by enriching the models with spaces, which was sufficient for the design context, as exact visual geometry was not the study’s focus.

Other enrichment tasks included site location and material properties via the interface, while internal gains, systems, and schedules were added during OSM creation. Integration of EnergyPlus (E+) and BuildWise2 (BW2) supported a dynamic inventory for electricity, heating, and water, but background processes relied on static data from ecoinvent.

The two non-domestic case studies revealed the approach’s dependence on BIM quality, highlighting the need for guidance on minimum standards. In practice, most energy models are built in proprietary tools like IES due to vendor support, so applying this architecture requires consultancies to have sufficient in-house expertise, which is not always available.

9. CONCLUSIONS

1. A key gap remains the lack of tightly coupled co-simulation approaches integrating Building Energy Simulation (BES) and Life Cycle Assessment (LCA). A review of the field showed that only one previous attempt has been made to achieve such integration, underlining the need for further development. The central question was: How can the tight integration of BES and LCA support building design and management?

2. To explore this, an open BIM-based co-simulation architecture combining EnergyPlus (E+) and BuildWise2 (BW2) was created and validated through two case studies in the UK and Luxembourg. Each building was assessed under three scenarios covering both construction and operation phases. The outcomes demonstrated the benefits of time-differentiated LCA over static assessments: environmental impacts can be tracked dynamically, enabling more accurate and granular evaluation. This makes it possible to determine precisely when and where interventions deliver the most significant effect.

3. The developed methodology for BES–LCA co-simulation advances the state of the art by providing a precise and repeatable process. For the industry, it serves as a practical guide for generating dynamic LCA results directly from BIM, reducing time and costs while supporting environmental targets. Validated through European case studies but applicable globally, the architecture enables researchers, practitioners, and policymakers to use it. Unlike static LCA, which only identifies what impacts occur, this approach also reveals when they are most significant, offering a more complete and holistic understanding of building behaviour.

10. REFERENCES

1. Alsharif R. A review on the challenges of BIM-based BEM automated application in AEC industry. 2019. <https://doi.org/10.13140/RG.2.2.26102.55366>.
2. Amissah P.K. Indoor air quality – combining air humidity with construction moisture. PhD thesis. Glasgow, United Kingdom: University of Strathclyde. 2005.
3. Cavalliere C, Habert G, Dell’Osso G.R, Hollberg A. Continuous BIM-based assessment of embodied environmental impacts throughout the design process. *J Clean Prod.* 2019. No. 3. P. 941–952. <https://doi.org/10.1016/j.jclepro.2018.11.247>.
4. Cellura M. et al. Modeling the energy and environmental life cycle of buildings: a co-simulation approach. *Renew Sustain Energy Rev.* 2017. No. 80. P. 733–742. <https://doi.org/10.1016/j.rser.2017.05.273>.

5. Elagiry M. et al. Copy IFC to Building Energy Performance Simulation: a systematic review of the main adopted tools and approaches. 2020.
6. Gomes C. et al. Co-simulation: state of the art. 2017. arXiv, p.arXiv:1702.00686.
7. Le D.V. et al. Deep reinforcement learning for tropical air free-cooled data center control. *ACM Trans Sens Netw.* 2021. No. 17(3). P. 1–28. <https://doi.org/10.1145/3439332>.
8. Mirsadeghi M. Co-simulation of building energy simulation and computational fluid dynamics for whole-building heat, air and moisture engineering. PhD Thesis. The Netherlands: Eindhoven University of Technology. 2011.
9. O'Donnell J.T. et al. Transforming BIM to BEM: generation of building geometry for the NASA ames sustainability base BIM. Lawrence Berkeley National Lab. 2013. <https://doi.org/10.2172/1168736>.
10. Page M.J. et al. The PRISMA 2020 statement: an updated guideline for reporting systematic reviews. *BMJ.* 2021. 372. No. 71. <https://doi.org/10.1136/bmj.n71>.
11. Pang X. et al. A framework for simulation-based real-time whole building performance assessment. *Build Environ.* 2012. No. 54. P. 100–108. <https://doi.org/10.1016/j.buildenv.2012.02.003>.
12. Sadeghipour Roudsar M. et al. Ladybug: a parametric environmental plugin for grasshopper to help designers create an environmentally-conscious design. In: 13th conference of international building performance simulation association. France: Chambéry; 2013.
13. Taveres-Cachat E. et al. Ten questions concerning co-simulation for performance prediction of advanced building envelopes. *Building and Environment.* 2021. No. 191. 107570. <https://doi.org/10.1016/j.buildenv.2020.107570>.
14. Trčka M. Co-simulation for performance prediction of innovative integrated mechanical energy systems in buildings. PhD Thesis. The Netherlands: Eindhoven University of Technology. 2008.
15. Yeung J. et al. An open building information modelling based co-simulation architecture to model building energy and environmental life cycle assessment: A case study on two buildings in the United Kingdom and Luxembourg. *Renew Sustain Ener Rev.* 2023. No. 183. 113419. <https://doi.org/10.1016/j.rser.2023.113419>.
16. Zhan S., Chong A. Data requirements and performance evaluation of model predictive control in buildings: a modeling perspective. *Renew Sustain Energy Rev.* 2021. No. 142. 110835. <https://doi.org/10.1016/j.rser.2021.110835>.

4. DEVELOPMENT OF 3D MODELS OF MICROCLIMATE FORMATION SYSTEMS OF PUBLIC BUILDINGS USING LINEAR SOFTWARE

Rybachov S. H., Moskvitina A. S.

1. Development of a BIM model of a building.

Building design is a complex, time-consuming and multi-factorial process involving a significant number of highly qualified specialists.

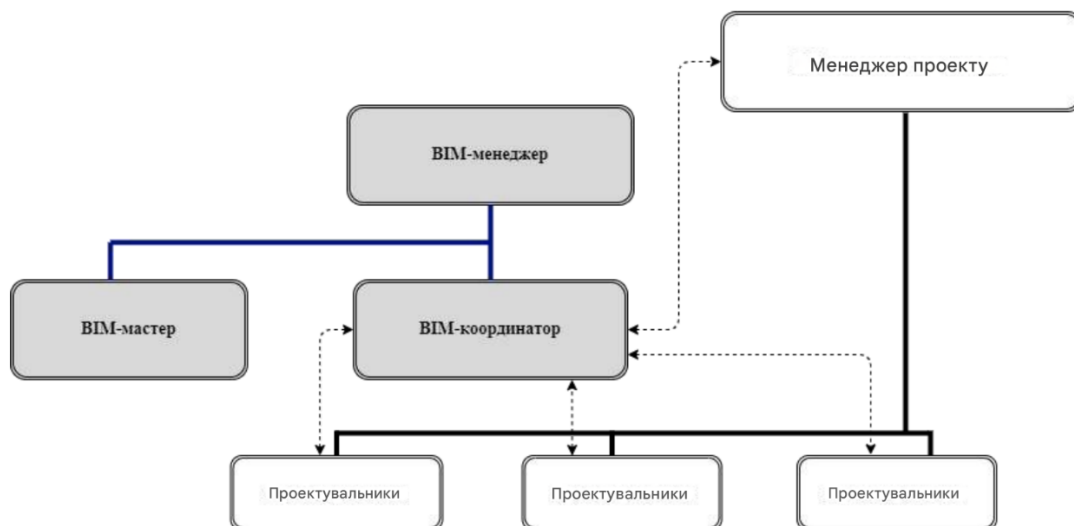


Figure 1-Diagram of interaction between participants of a BIM project

Everyone who participates should clearly understand their task and responsibilities in the organization. Job titles are regulated within the organization, so there may also be differences. For example, according to the IT industry, the development of a specialist from a beginner to a senior specialist includes several stages: trainee (intern) - obtaining basic knowledge and skills under the guidance of experienced colleagues; junior specialist-performing simple tasks independently, gradually gaining experience; middle specialist-independently performing more complex tasks, starting to participate in large projects; senior - full responsibility for major projects, mentoring junior colleagues, and making strategic decisions.

Employees who perform direct design work can also be called differently, but most often the ciphers of documentation sections are used as a suffix to the position of an engineer, such as a designer of S, VC or AR, or in English HVAC, WSS, AR, etc.

1.1 stages of developing a digital information model (BIM)

In this paper, we consider the features of the development of the OV or HVAC section. You should consider such important stages of the entire workflow based on the example project (Figure 2).

The architect receives a technical task, creates a basic model of the building, and performs a preliminary check of his section for gross errors. Then the model is passed to the following departments: architectural and construction, plumbing, etc. design engineers develop their sections using the basic model of the structure as a basis, and also check the results of filling the structure with their own systems.

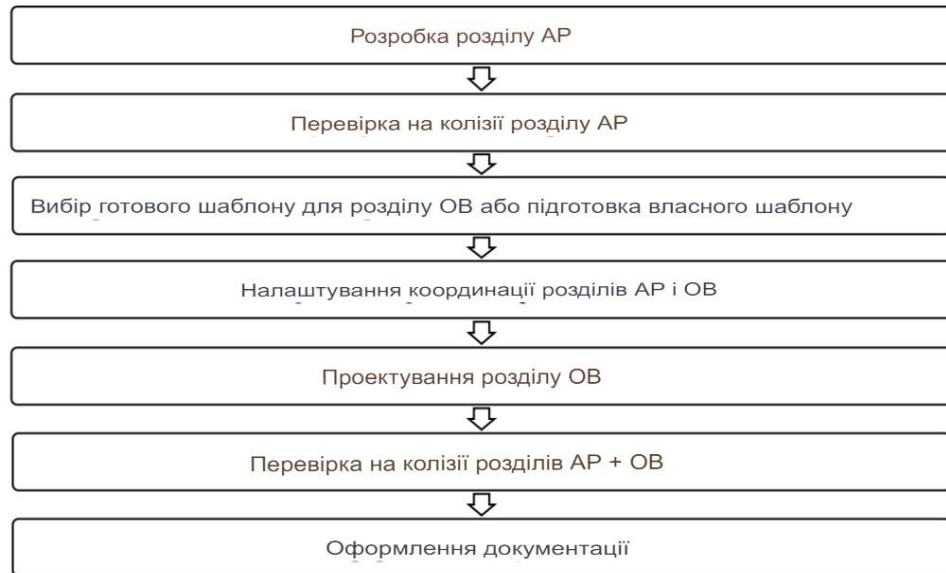


Figure 2-main stages of project development

Валідація Model validation

The use of information modeling technologies provides a number of advantages for developers of engineering systems:

- collaboration allows you to develop a single project in a shared data environment by relevant specialists, even with remote access.
- the ability to monitor elements of adjacent sections and receive notifications about changes made to them;
- availability of automated marking tools for object models.
- automatic generation of system and equipment element specifications.

A consolidated digital model is a digital information model of an object that consists of separate digital information models and engineering digital terrain models (for example, from different sections or parts of a construction object), combined in such a way that making changes to one model does not lead to changes in the others. The main purpose of the summary model is to support the processes of coordinating technical solutions and identifying collisions.

After drawing up a summary model from the developed versions of sections, you need to check again, on the basis of which a report is generated indicating all comments and violations. Making corrections and changes within a certain time frame is assigned to the engineers from the corresponding section, and only when all the shortcomings are agreed and corrected do they complete the documentation and transfer it to the site.

In organizations with a large staff, designers from different departments can communicate directly with each other, and the VIM Coordinator (VIM Coordinator) is the link. The DSTU ISO / TS 12911: 2020 standard specifies his professional actions (Fig. 3).

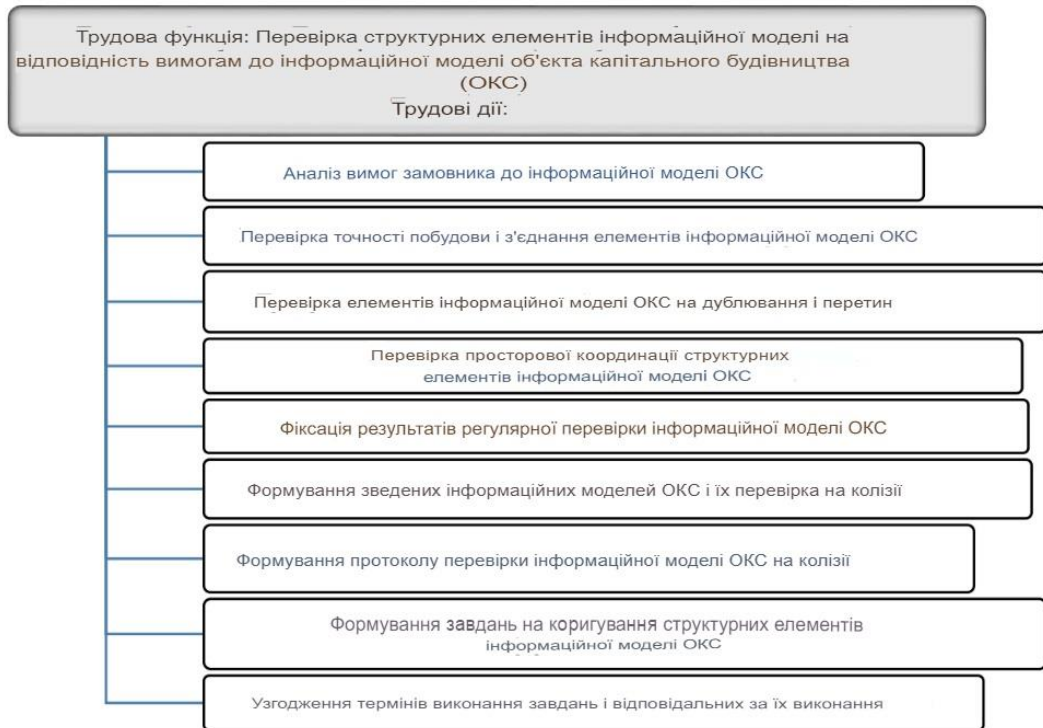


Figure 3-Responsibilities of the VIM Coordinator

The term "collision", borrowed from foreign languages, is widely used in various industries. In the context of construction design, it means an error that occurs due to a mismatch or application between adjacent parts of the same project, as well as due to the intersection of the design boundaries of individual objects.

Identification and elimination of such conflicts at the stage of project development under proper control allows you to reduce the amount of further improvements and labor resources.

One of these main reasons why there are conflicts in the design process is the involvement of large numbers of specialists, who usually work separately or remotely. This includes coordination between participants and based on the fact that error detection and model validation take longer to complete than during the design phase itself. As the number of specialists involved increases, it also increases the likelihood of inaccuracies and errors in the project.

Based on practice, there are three main groups of collisions:

- hard-physical cross-section of elements in the model.
- soft-elements of the model geometry are not connected to each other, but the surrounding space and the zones of normalized values for operation intersect.
- spatio-temporal-the intersection of elements taking into account the time factor estimated on the basis of calendar or network graphs.

To develop the project, we used the Autodesk Revit software package, which is currently the most popular in the design environment based on information modeling technologies. You can use various software tools to perform the intersection check process. For example, you can check the physical intersection of elements with the built-in Autodesk Revit tools

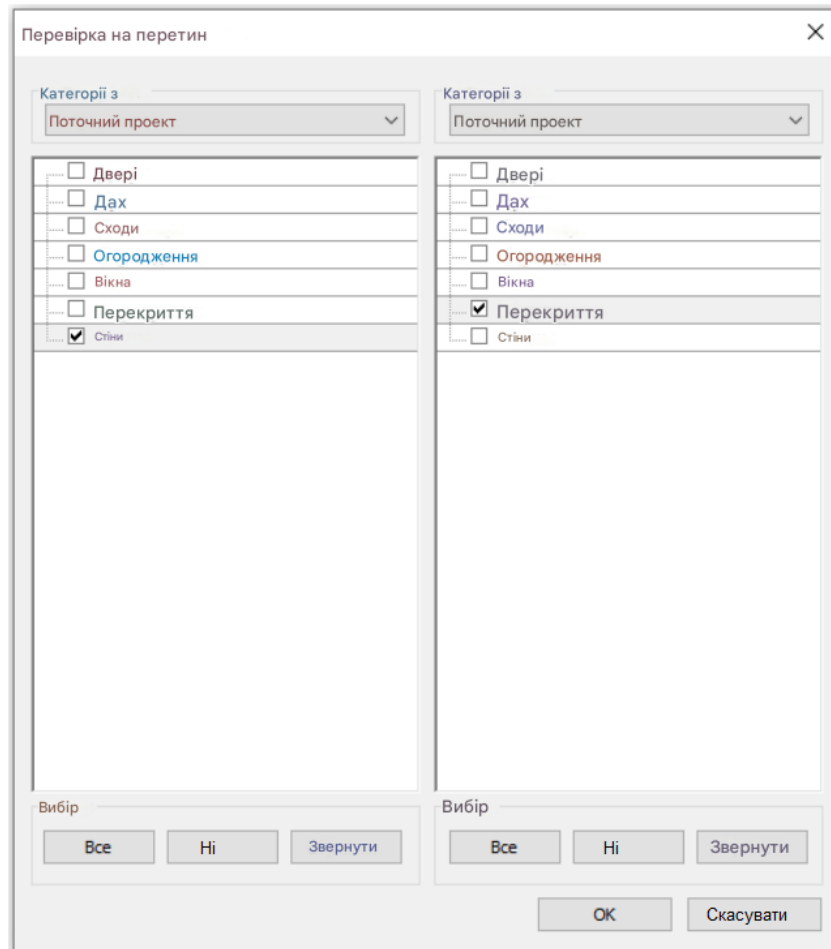


Figure 4-Configuring cross-section validation parameters using Autodesk Revit

The result of the check is a dialog box with a message. If errors are detected, they will be listed as an active list, where you can find and view each incorrect situation (Figure 5).

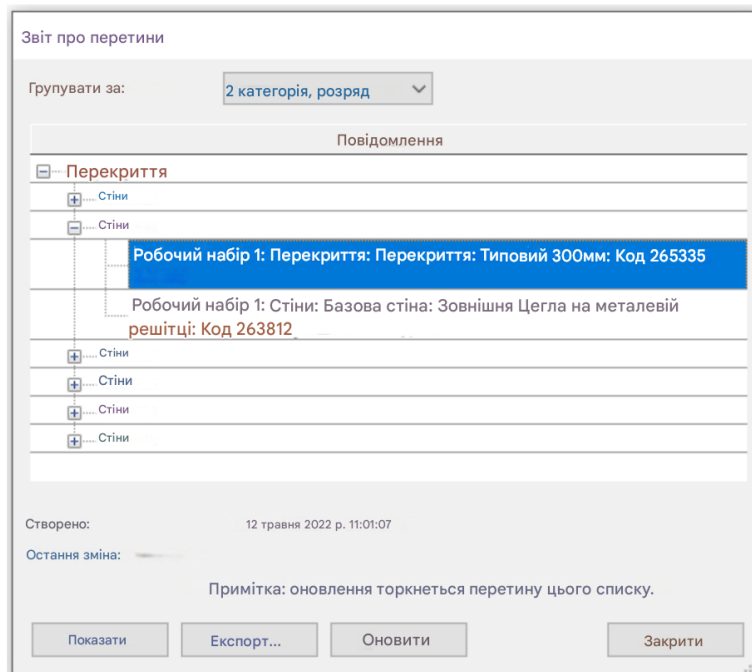


Figure 5-Verification report

However, this option is not always convenient to use. When you need to check the summary model, most often check it in another program. For example, in Autodesk Navisworks, where the model is built.

1.2 Collaboration kit

A workflow that involves interaction between project participants is called collaboration. This approach can solve a common problem or the work of one specialist that will be used in the work of another.

When working together, it is important to maintain the presence of linked networks in the working file for a one-time build with elements already placed in the project. To connect files, use the **Link Manager tool** on the Manage tab (Figure 6).

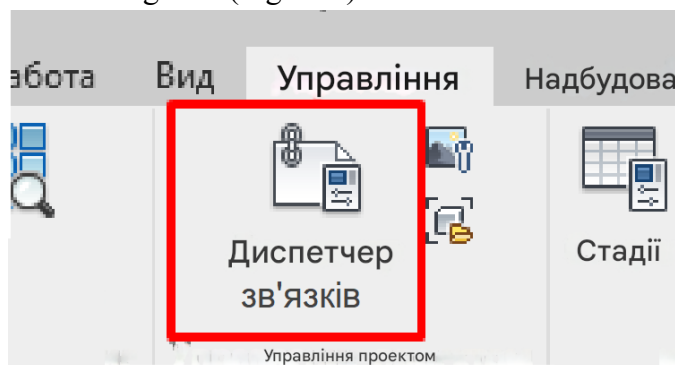


Figure 6-Link Manager tool

The dialog box will display files that have already been attached, and you can connect the document using the **Add button** (Figure 7).

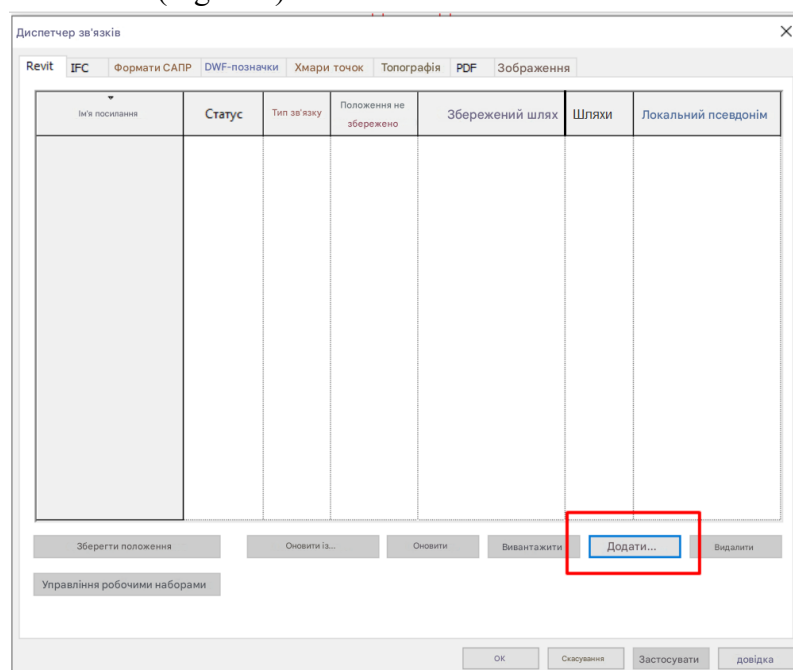


Figure 7-Link Manager dialog box

Then you need to specify the location of the file with the adjacent section by going to the appropriate folder. After selecting it, make sure to perform preliminary loading settings by setting the placement parameter to **Auto-by common coordinates** (Fig. 8).

If you select **Set** before uploading a file, a dialog box will appear with a list of worksets that you can close to prevent them from being uploaded to the project. It is recommended to close the **general levels and grids working set** (Figure 9).

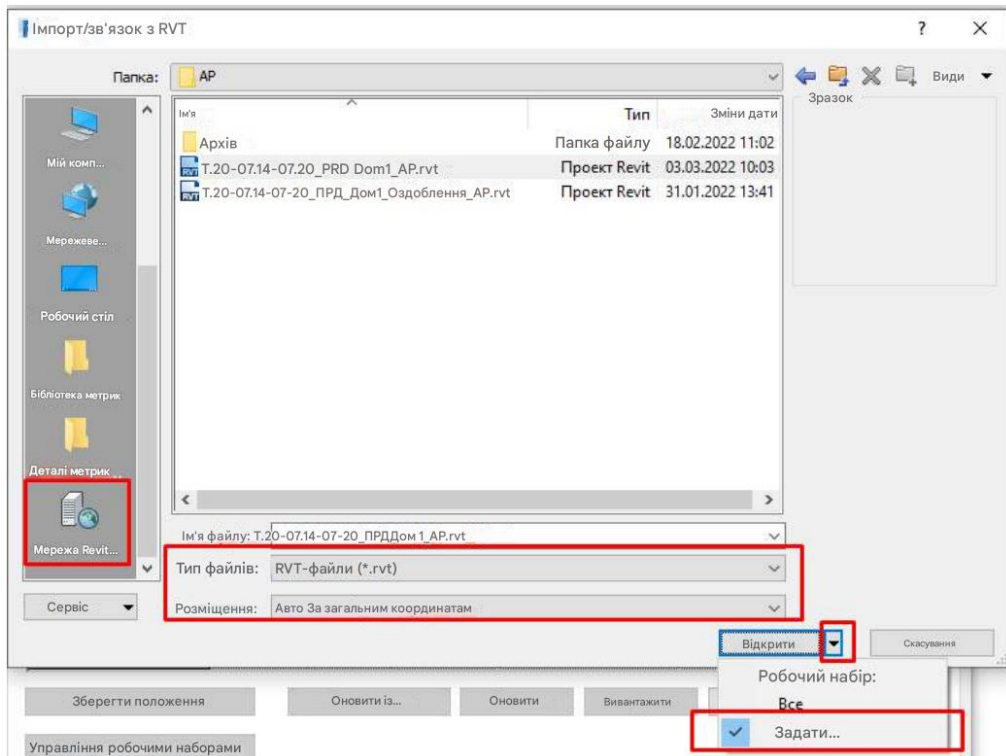


Figure 8 - Configuring communication parameters

To start the design stage of engineering systems, a digital model of the building was prepared (Figure. 10).

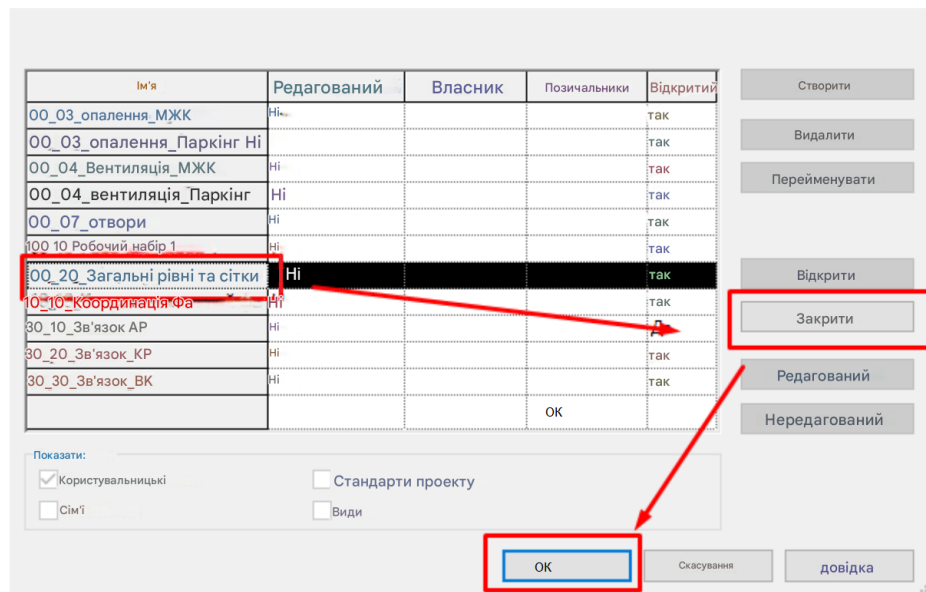


Figure 9-Disabling the working set

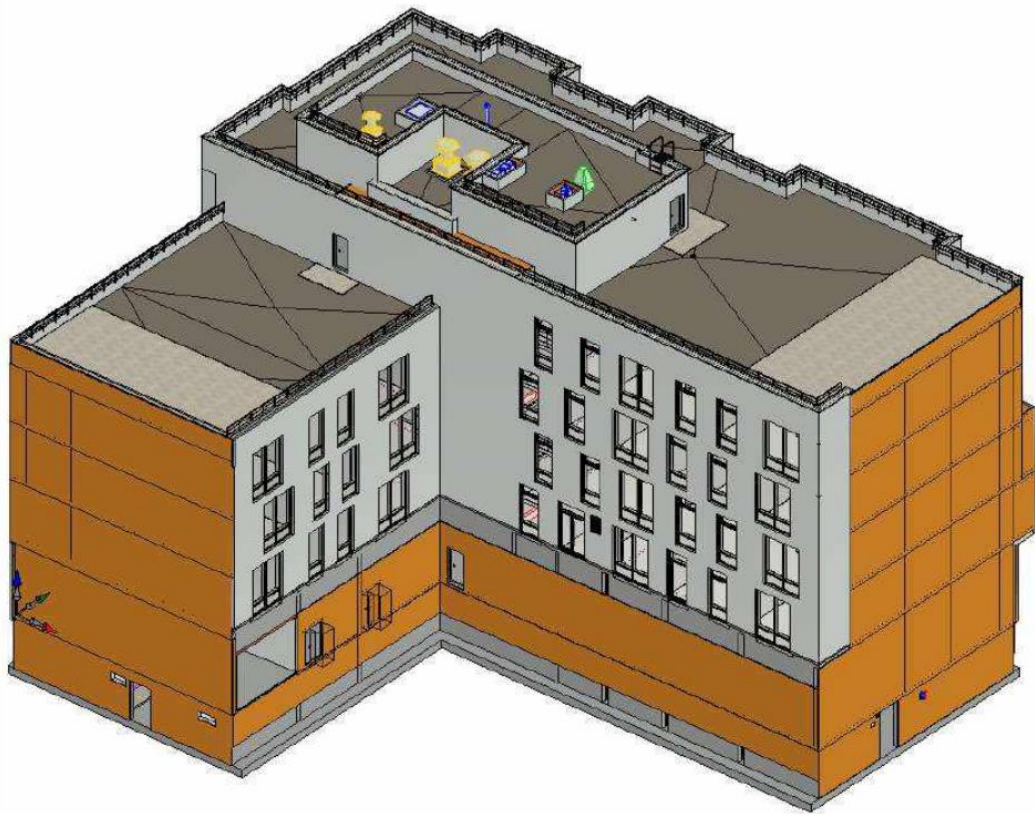


Figure 10-digital model of the building

2. DESIGNING AN OV PARTITION WITH AUTODESK REVIT

Autodesk is constantly improving templates for project development, this is due to the release of new versions of programs, the development of the functionality and capabilities of A08K programs-templates for them have been used since 2016, which can already be recognized as an industry standard, since they comply with EN, DIN, DSTU.

A template is a pre-configured project environment with a set of elements for modeling, fonts, annotations, custom specifications, and other parameters [20].

According to various sources, from 60 to 80% of designers use ADSK: templates or ADSK parameters in their work. The template can be downloaded from the website of the company BIM2B [21], whose experts participated in the development of templates and standards.

Included with adsk templates for version 2021 is BIM2B the VIM2B Templates Tools plugin from this company. This application allows you to automate the actions of designers. The unique feature of this version of the plugin is the addition of all engineering sections [21].

Configuring the trace

Before you start designing the OS section, you need to check and correct the specified trace parameters if necessary. This will allow you not to completely change the pipe materials for each system, especially since you can no longer edit the thickness values during operation. To check the set values, use **the HVAC System configuration tool OBiKon the management tab** (Figure 11).

In the menu, you can view the settings for track designations, their standard sizes (dimensions, thicknesses, etc.), slopes, and other parameters.

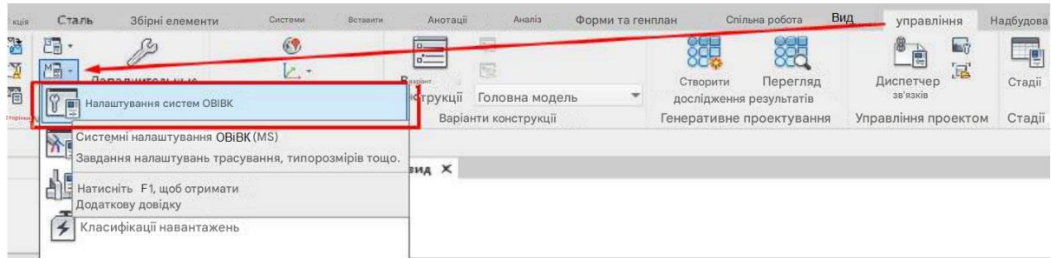


Figure 11-Tool for configuring trace parameters

When you select **Standard sizes** (pis. 12), you can check the dimensions of the specified networks.

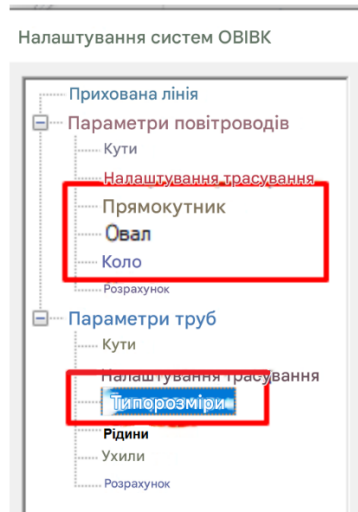


Fig. 12-Item Standard Sizes

You can edit data (this applies only to the text part), create new values, or delete redundant values. You cannot edit existing dimensions in the dimensions catalog (pis 13). We recommend that you check if all the required dimensions are listed. If there is a lack of data, you must create and pass the task to the VIM coordinator.

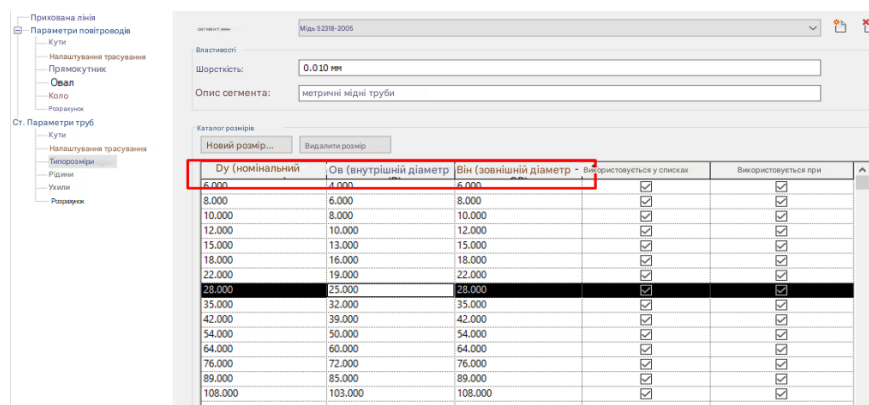


Figure 13-Size catalog

2.1 Features and recommendations for working with plans

Before building networks, you need to create floor plans for the corresponding floors. You can create a plan using **the Views in plan command on the View tab** (Figure 14).

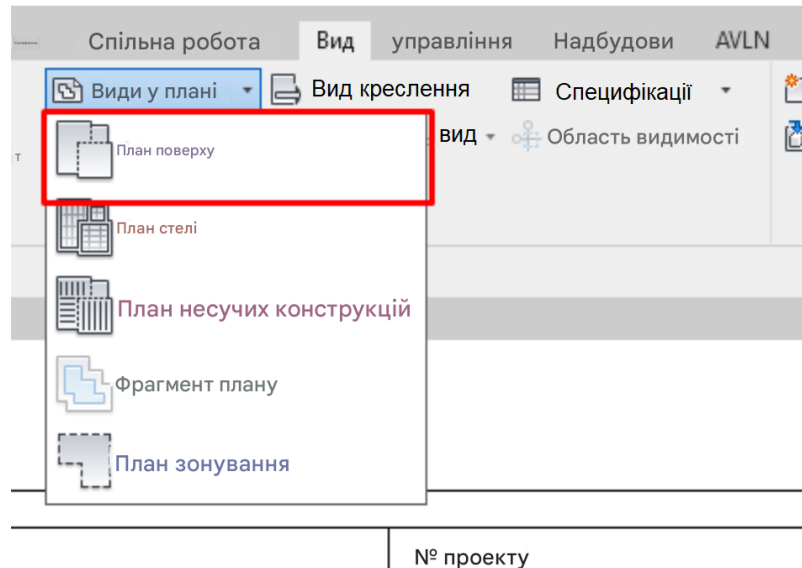


Figure 14 - tool for creating a floor plan

In the dialog box that appears, you need to select the plan category. Depending on this, the suggested list will be filtered out and unnecessary elements will be hidden (psee figure 15). By unchecking **the Do not copy existing views** option, you can quickly create multiple floor plans for the same floor without additional copying.

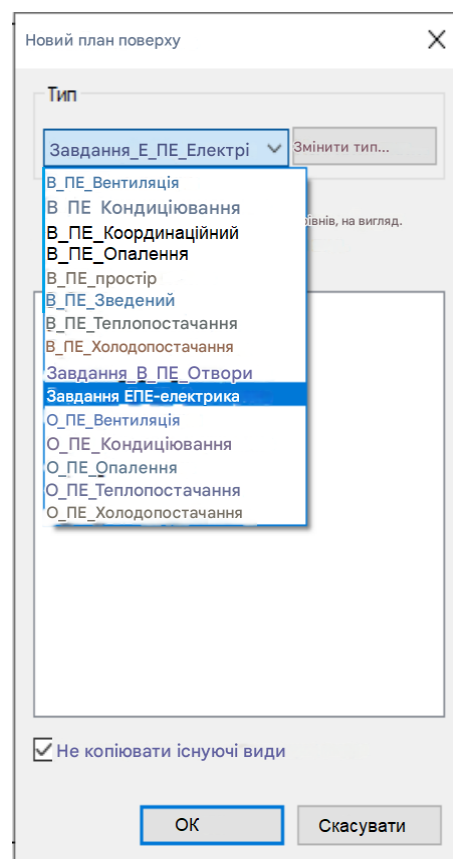


Figure 15-Selecting the category of the new floor plan that is being created.

In the plan category, it is common to use the prefixes: B, tasks, O. Depending on the selected category, plans created automatically are divided into groups in the Project Manager: **auxiliary views, tasks, and decorated views.**

Assigning prefixes:

B- these are auxiliary plans that can be used to enable the display of other networks, configure the range, and other parameters. This is a flexibly configured category of plans in the project, as it allows you to make the appearance more convenient for viewing and coordinating in your section. In **the Project Manager**, all created plans fall into **the auxiliary views category** (pis. 16).

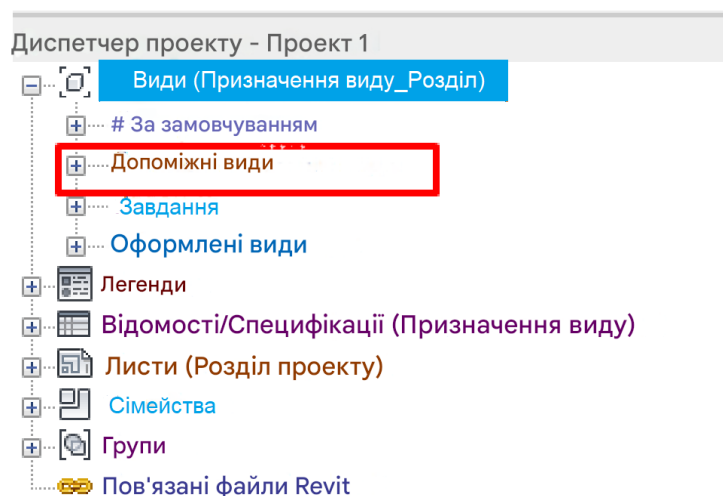


Figure 16-Auxiliary views category in **the Project Manager**

Task-this category of plans is intended for completing tasks for the electricity section or tasks for holes (depending on the selected category).

O-these are designed views that are later used to display data on a sheet of paper only for output.

We recommend renaming the newly created plan view according to the category, since by default the view name does not have a prefix. Specifying the appropriate category will significantly simplify orientation in **the Project Manager** (Figure 17).



Figure 17-Renaming the plan

We recommend that you change the plan category under certain circumstances. For example, if the auxiliary plan is suitable for displaying on a sheet, you can move it to the **designed views category**.

Views are sorted in the Project Manager based on the following parameters: **Adsk_specification of the view** (value options: **auxiliary views, decorated views, tasks**, etc.) and **ADSK_SSTAMP project section** (value options: **ventilation, heating**, etc.) (Figure 18).

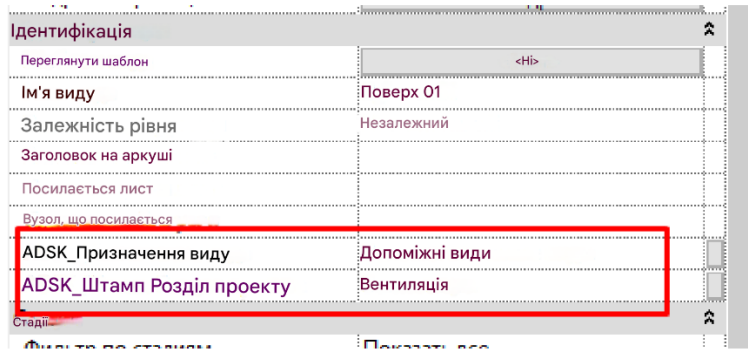


Figure 18-View parameters in the **Properties panel**

Plan types can be copied in the Project Manager via the context menu by selecting one of two options (psee figure 19):

- 1) **copy** - without markings, dimensions, or other annotation symbols.
- 2) **copy with detail** – a full-fledged copy of the plan with stamps and dimensions.

A copy of the view will appear automatically in **the project manager**. By default, a number will be added to the name of the original name at the end, so it is important to immediately change the name of the created copy.

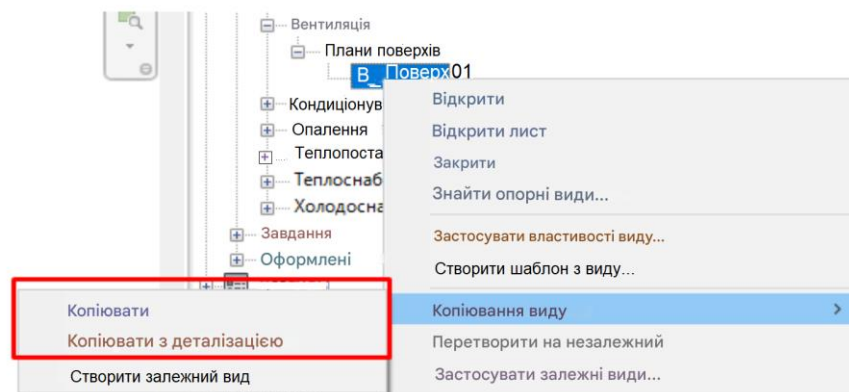


Figure 19-Copying a view in **the project manager**

2.2 basic principles of building a heating system

The main elements of the heating system development are: pipes, pipeline fittings, equipment, pipeline insulation. Let's consider the features of the formation of these elements.

3.2.1 recommendations for building pipeline networks

Before designing, you must perform the following preparatory steps:

- 1) create an appropriate heating plan.
- 2) setting up heating systems.

To adjust the parameters of heating systems in the Project Manager in the family folder, find the category pipeline systems (pis. 20).

This folder contains the system categories that the project has (Figure 21). Using the recommended template, you should only use systems with the ADSK prefix.

When you activate the Pipe tool, on the Systems tab (Figure 22), you can select system data in the Properties panel (Figure 23).

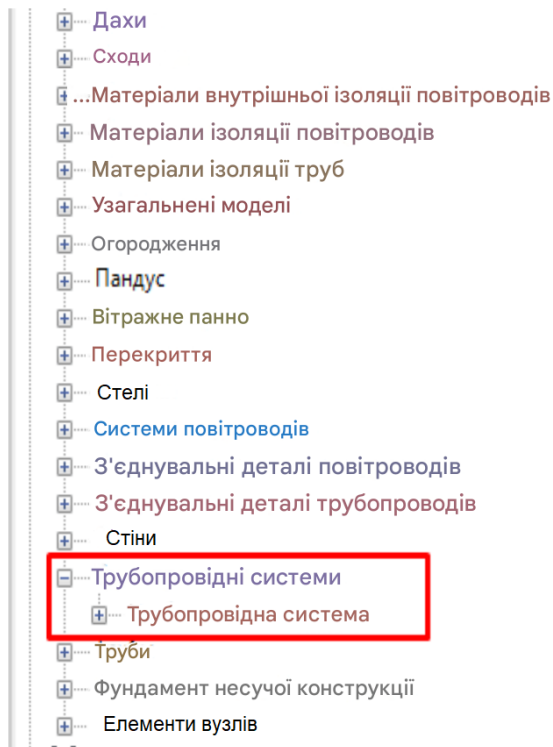


Figure 20-Pipeline Systems folder in the Project Manager

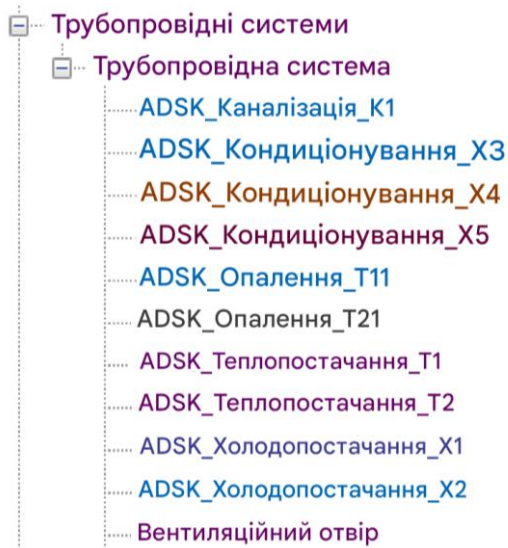


Figure 21-Project system categories

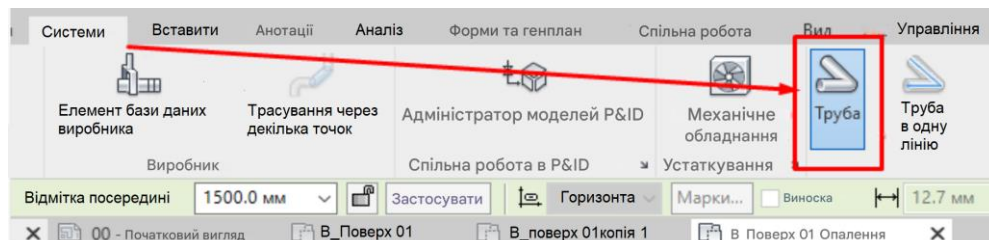


Fig. 22-Pipe Tool

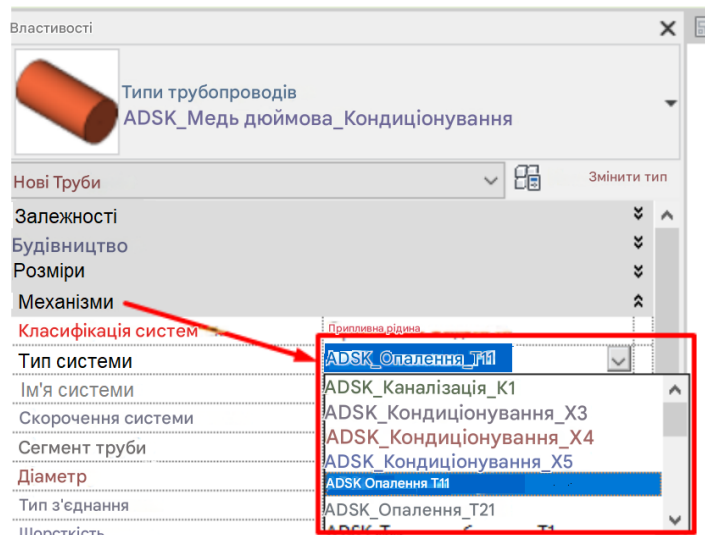


Figure 23 - Selecting the system type in the **Properties** panel

If you need to add a new system, such as the T11 system with the suffix 1, you need to make a copy of the ADSK system in the Project Manager_Opalenny_TP_ (ris. 24).

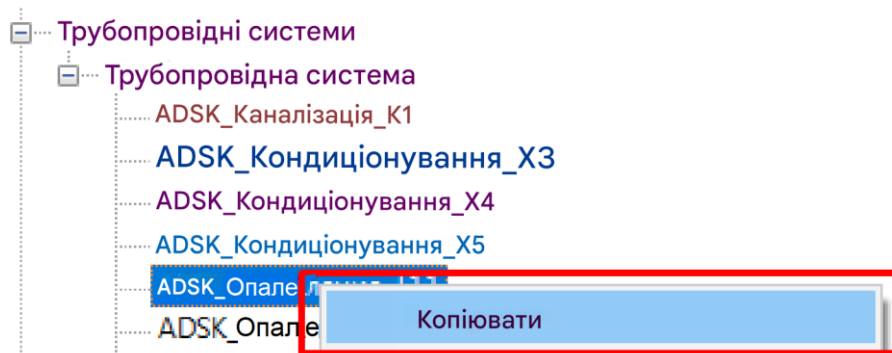


Figure 24-Copying the system

You need to rename the new system type because the number 2 will automatically appear at the end of the name (pis. 25).

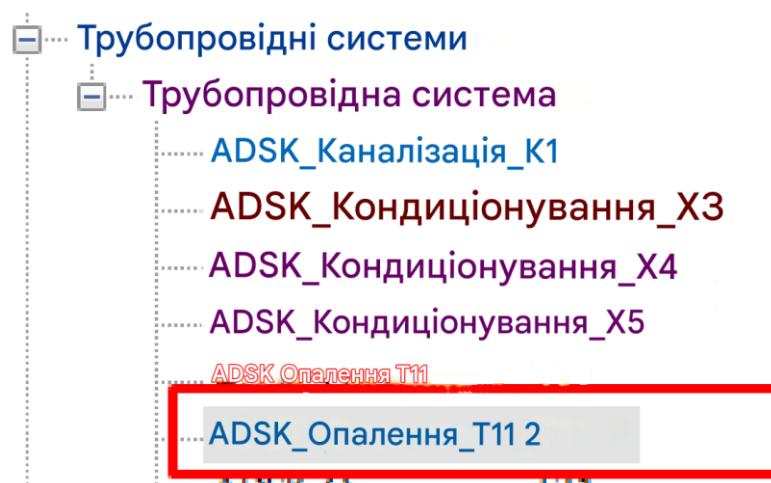


Figure 25-new system

You can rename it using the command from the context menu: to avoid duplicating abbreviations for the system, you need to change it in the new type.

In the dialog box that opens, you can configure various parameters, such as color, temperature, and abbreviations for the system (see figure 26). If the marking plan is **T11. 1**, then the **abbreviation parameter** has the same value.

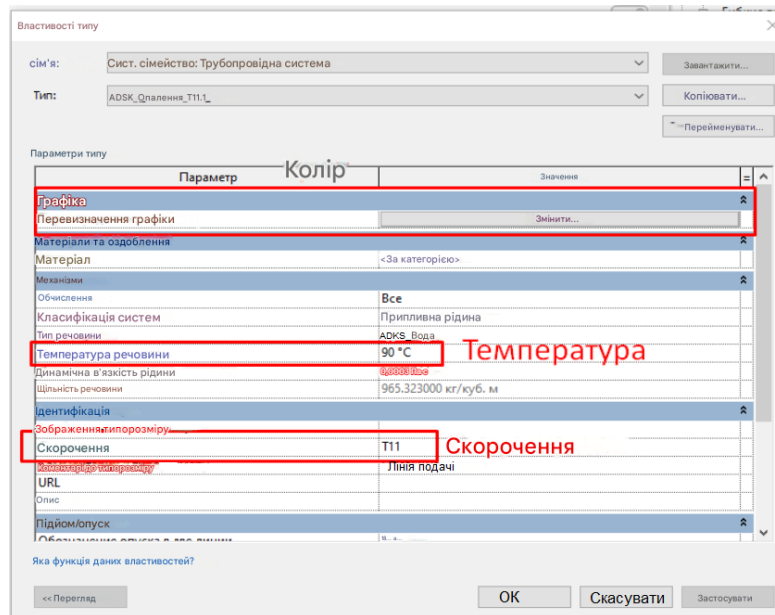


Figure 26-Type Properties dialog box

Now, when building, you can select a new system that has the necessary abbreviation specified in advance (Revit often uses the system abbreviation as the system name) (Figure 27).

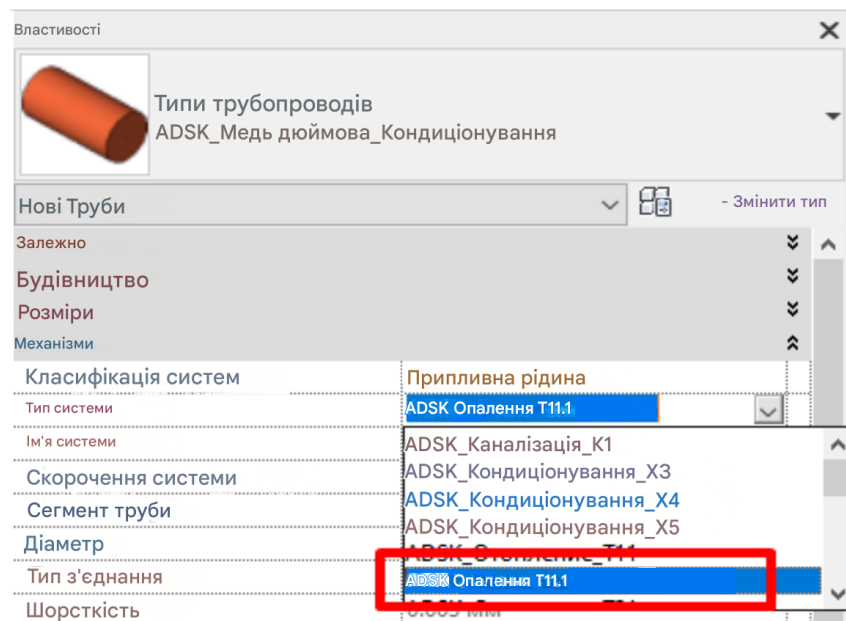


Figure 27-Selecting a new system type

After these settings, you can proceed to selecting the pipeline type for design by activating **the Pipe tool on the Systems tab**. Parameters of pipeline types will be available for viewing and selecting in the Properties panel (pic. 28). All types loaded in the project can be viewed in the drop-down list.

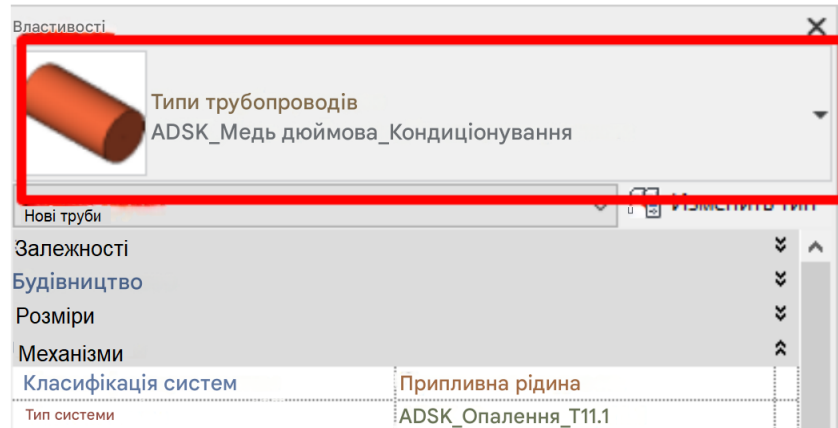


Figure 28-types of pipelines available in the project

After the selected type, the design process begins, the principle of which is similar to the construction method in the AutoCAD program. You must check the following parameters before specifying the location:

- the level reference value must correspond to the level of the plan where the pipeline is planned to be built (Figure 29).

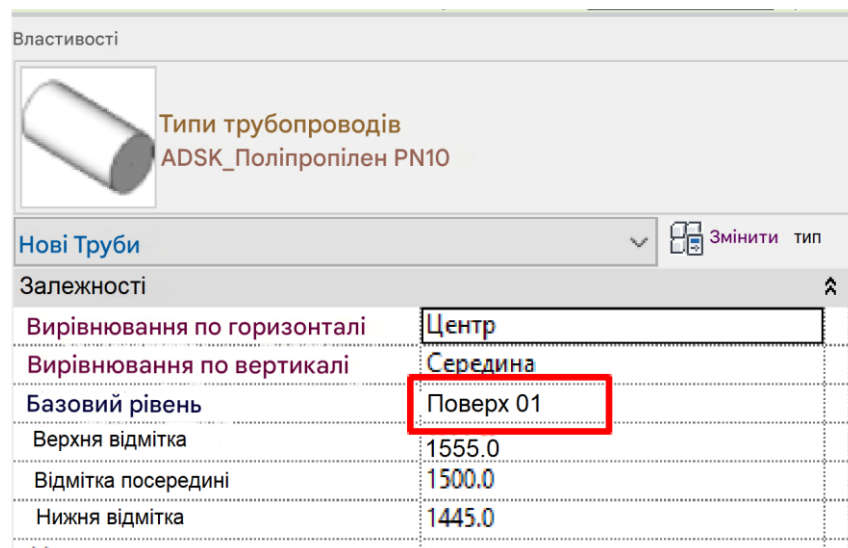


Figure 29 - Setting up the base level

- the value of the marker is indicated relative to the level at which the plan is located (pis. 30). You can also choose from which section of the pipe the distance will be measured. For example, if the first floor is located at 0.800 and construction is planned for the mid-1500 mark, then the pipeline will be located at a height of 2300 from the conditional 0.000.

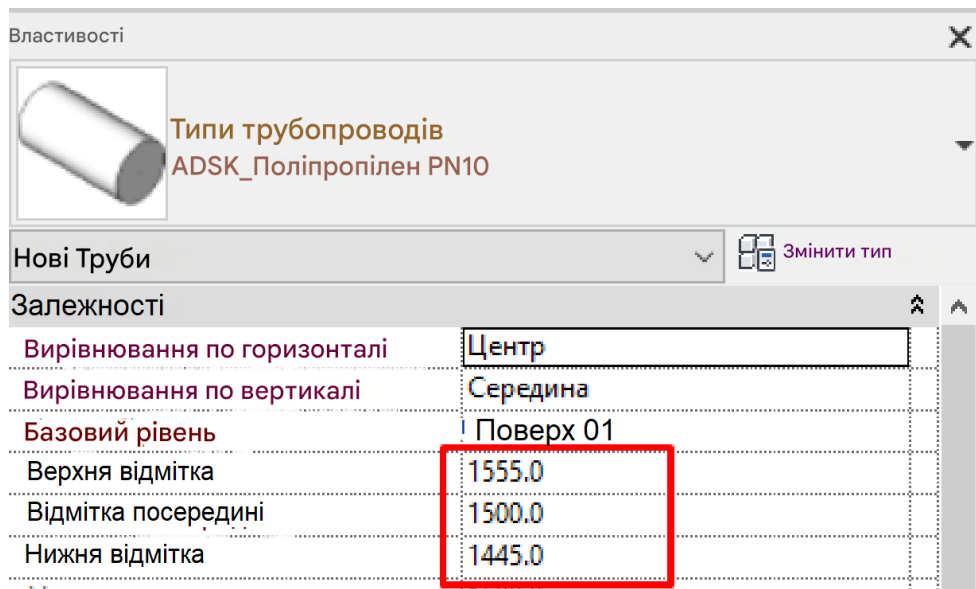


Figure 30 - Setting placemark values

- the system type must be selected correctly.
- the tilt value (Figure. 31) should only be adjusted if the customer requires it. It is not recommended to use this parameter when building heating networks, since the slopes in this case are minimal, and, therefore, it is better to design networks without a slope.

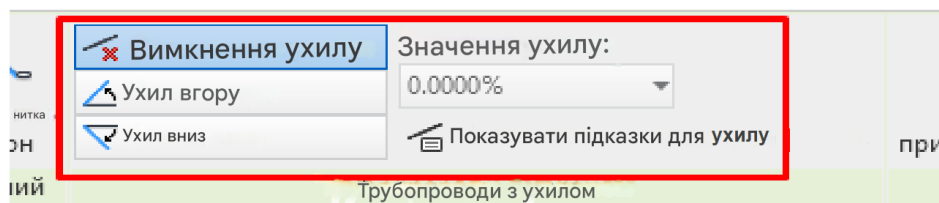


Figure 31 - Setting the slope value

Variants of slope values:

- 1) slope up – from the starting point of construction, the pipe will go up by the specified slope value.
- 2) slope down - from the starting point of construction, the pipe will go down by the specified slope value.

Features when building pipeline networks.

It is important to know about some nuances that are often encountered in the work:

- There are warnings in the fitting families that will show you that the fitting angle is not correct (Figure 32).

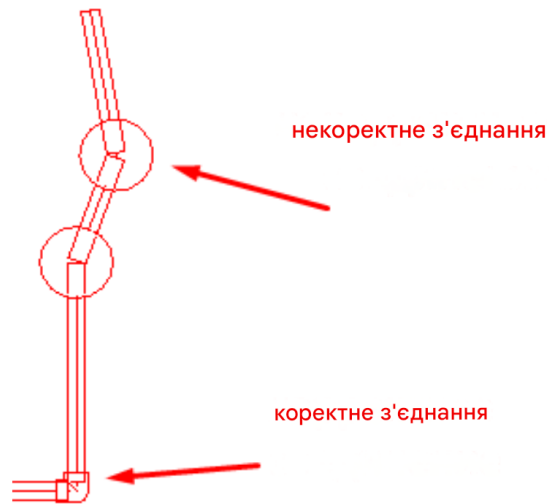


Fig. 32-Pipe connection options

To build risers, it is enough to select a pipeline, specify the starting point of construction, and the pipe is not considered built (pis. 33). Then change the binding and confirm by double-clicking the **Apply button** (pis. 34). As a result of these manipulations, the construction is finished.

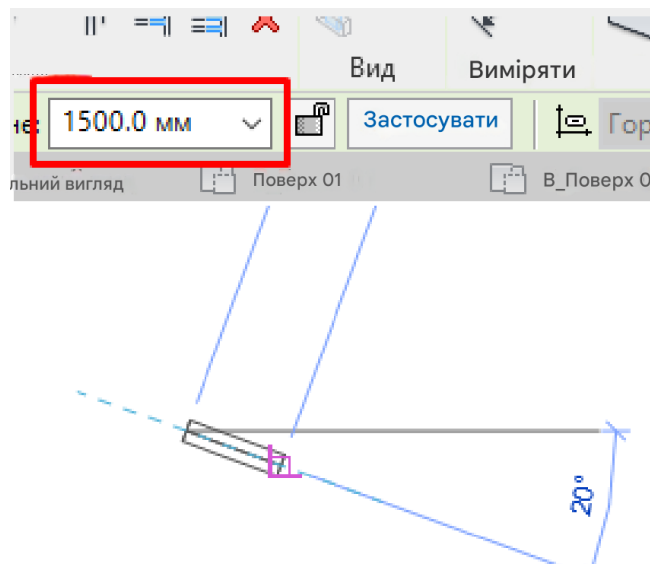


Figure 33-indication of the starting point of construction

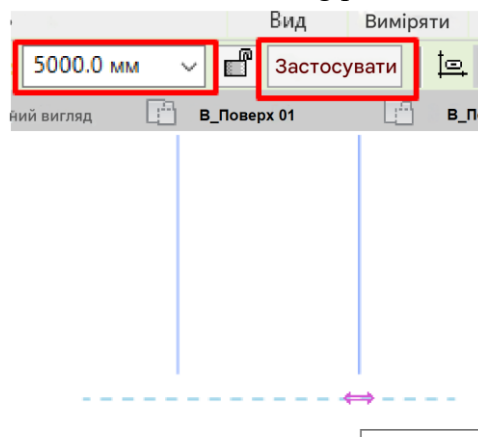


Figure 34 - changing the binding value and confirming the action

- The most common error that pops up when building networks is "the direction of the duct/pipe has been reversed, which caused connection errors" (Figure 35).

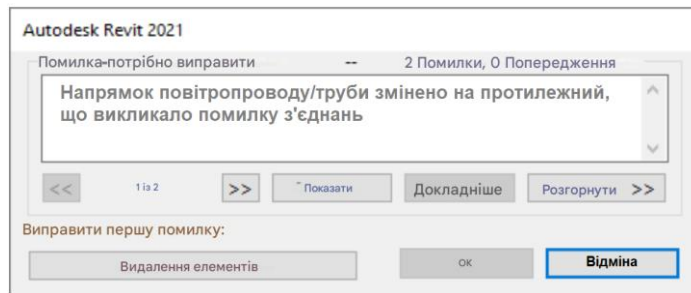


Figure 35-Pop-up error message

Let's take an example of this situation. Let's say that a U-shaped compensator has been created, and it was decided to change its dimensions by reducing the height (pis. 36).

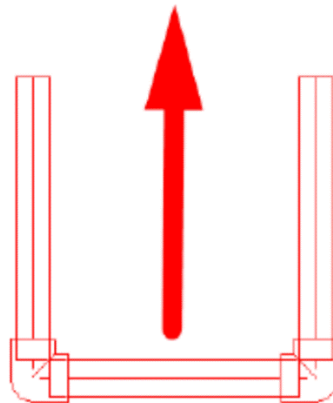


Fig. 36-U-shaped expansion joint

When the pipe is shifted, the program will show the previously presented error that the pipe has turned in the opposite direction, but the fittings will not automatically turn (pis. 37).

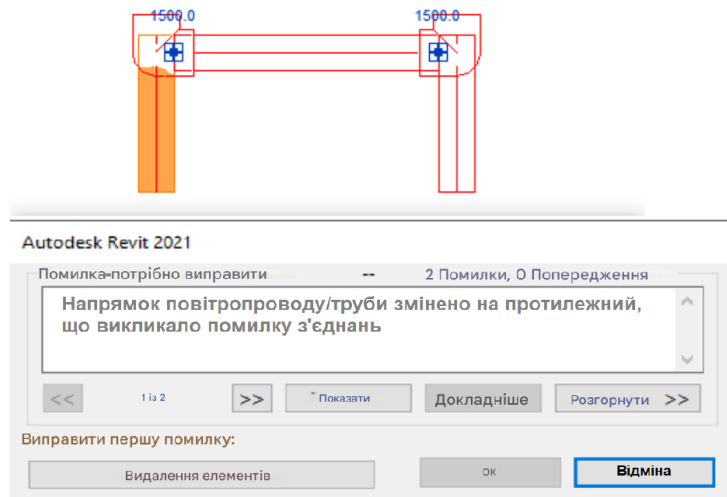


Figure 37-Error situation

- Keep in mind that when using the plugin from B1M2B, the Adsk_assembly and Adsk_quality parameters **ta in the pipe properties are automatically filled in** (Fig. 38). This will help you fill out the hardware specifications later.

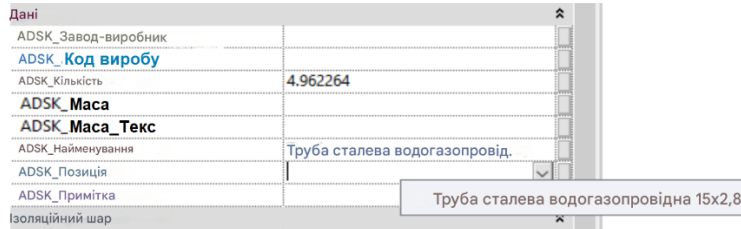


Figure 38-Filled parameters Adsk_name

2.2.1 Recommendations for building rebar structures

You need to load the main valve families into the project beforehand, because there are no such elements in the template. Upload tasks must be compiled and sent to the VIM Coordinator. The location of fittings should be dealt with after the stage of building primary networks.

The Pipeline Fittings tool is located on the Systems tab.

After activating the tool, the list of types loaded into the project will become available (Figure 39). You must select the type that corresponds to the pipeline diameter. Placement is performed by pointing the centerline of the pipe with the mouse (Fig. 40).

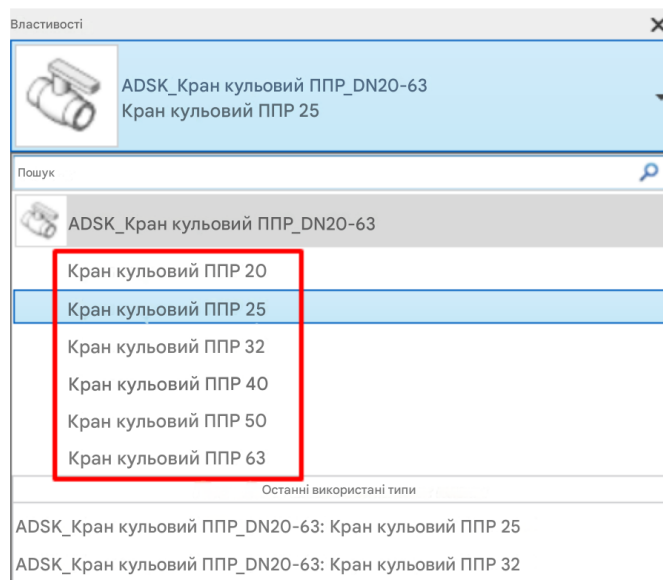


Figure 39-types of pipeline fittings

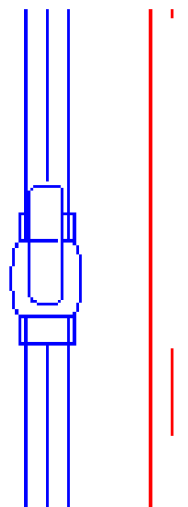


Fig. 40-placement on the pipe

It is important to note some features:

- It is possible to build pipelines from the fittings placed in the design industry. To do this, just click on the crosshair that appears when the element is placed (pis. 41).



Fig. 41-Construction of the pipeline from the valve

- To avoid intersections with other networks, you can change the direction of the valve and turn it at a certain angle.

- When the anchor level is changed, the marker is automatically recalculated.

2.2.2 Recommendations for working with the equipment category

The category of equipment in the S section includes heat appliances: convectors, radiators, etc. In order to arrange the equipment, go to the **systems** tab and select the category **mechanical equipment**.

Next, the equipment placement stage is performed with setting up binding by level and offset (ris. 42).

The ADSK21 template includes a universal radiator with various additional connection properties (pis. 43).

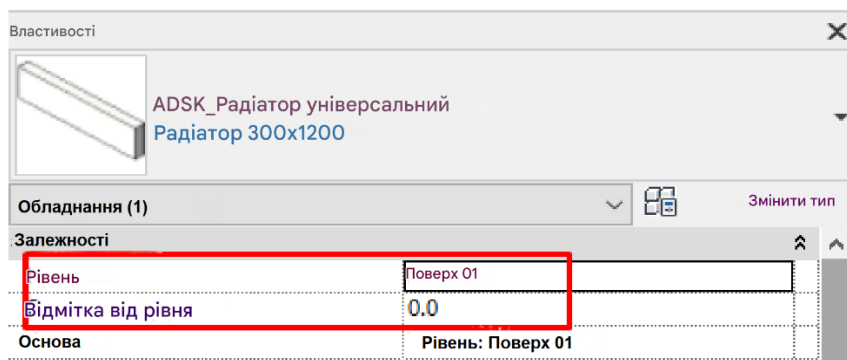


Figure 42-Setting the level and marker values

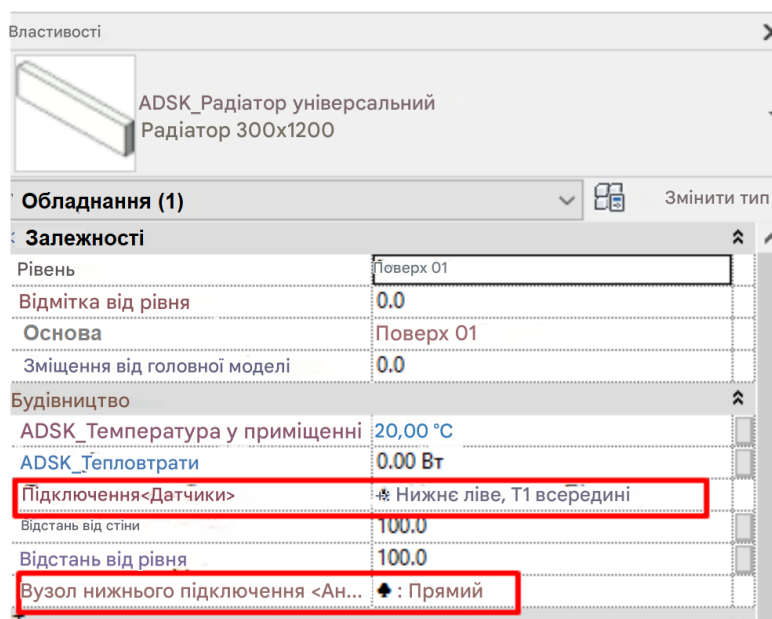


Figure 43-Additional properties of the universal radiator

After placing the radiator, you must connect it.

1) align the pipes to the connection points (fig. 44). For this step, we recommend using **the Align tool**.

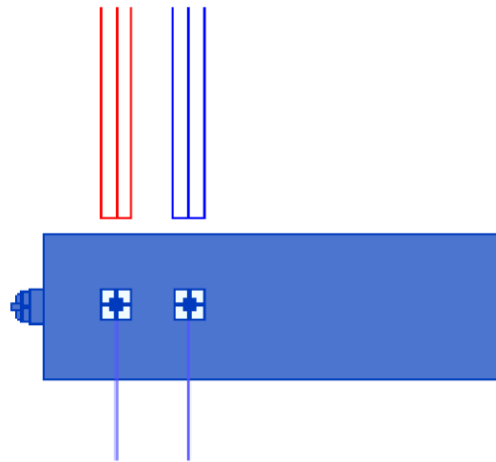


Figure 44-Point alignment

- 2) specify the connection axis (use the Tab key for convenienceTab) (pis. 45).
- 3) Select the pipe axis, considering that it is located on the same axis as the connection.

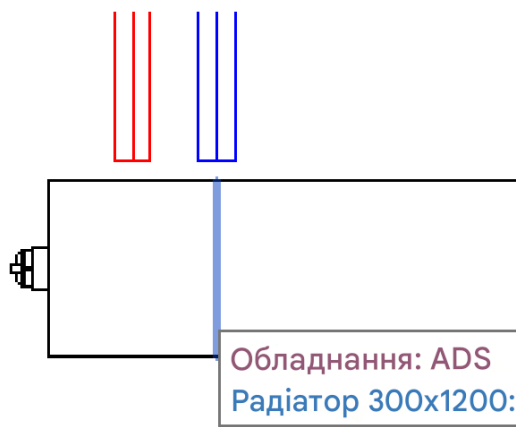


Figure 45-Defining the centerline

4) Making the connection. When the pipe continues to the equipment, an identification mark (square or cross circle) should appear, which means that the connection can be made (pis. 46).

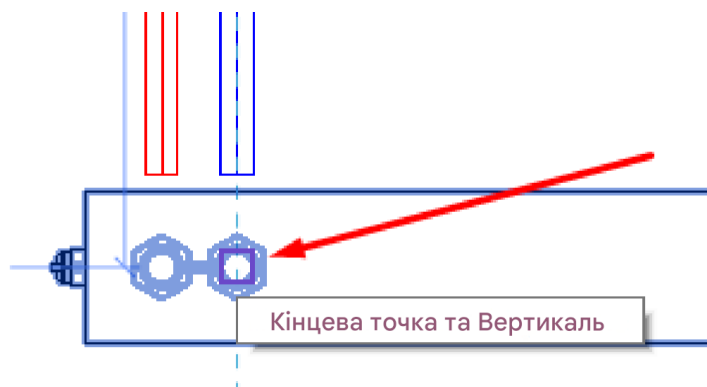


Fig. 46 - connecting the pipe to the radiator

Figure p 47 shows the connection result.

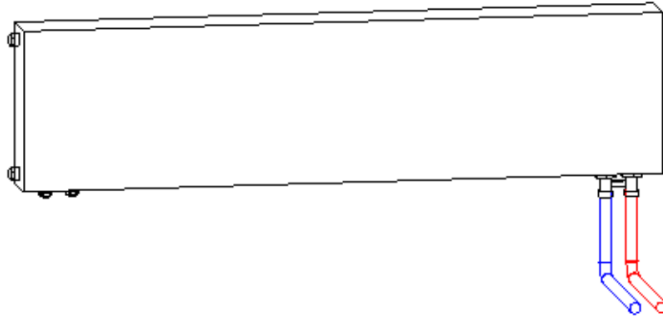


Figure 47-Connection result

You should pay attention to some features:

- Unlike pipelines and fittings, equipment will automatically move to another level if you change the level reference.
- Equipment can have multiple system names specified, this is a feature of the category (Figure 48). It all depends on the number of connections.

Механізми	
ADSK_Налаштування клапана	0,000000
ADSK_Втрата тиску рідини	0,000000 Па
Радіатор_Втрати тиску	0,000000 Па
Клапан_Втрати тиску	0,000000 Па
Клапан_Витрата рідини	0,00 л/с
Класифікація систем	Зворотна рідина, ...
Ім'я системи	T11, T21 1
Механізми Витрата	

Figure 48-Multiple values for the system name property

To make it easier to connect equipment, you can use auxiliary views: sections, a fragmented view that can be performed using **the Selection Frame tool**.

2.2.3 Recommendations for placing insulation

Adding insulation to a pipeline can be done in two ways: on a plan/ 3D view, or by selecting pipelines from the specification. Both options are equally effective, but the application depends on the situation. Let's look at each of them.

Location of insulation on a plan or 3D view

The placement option on the plan or 3D view is almost identical. It all depends on the ease of selection and the number of necessary objects. To implement this method, we recommend following these steps:

- 1) select the necessary pipelines using the selection box or Tab keyTab.
- 2) activate **the add insulation tool** (fig. 49).

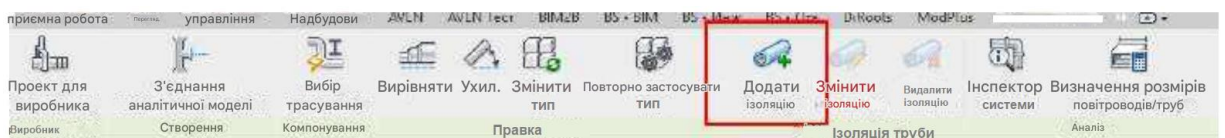


Figure 49-Add insulation tool

- 3) in the dialog box that appears, select the insulation type and set the thickness value (pis. 50).

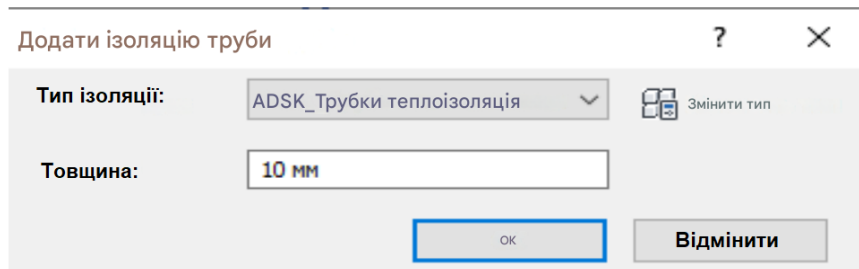


Figure 50-Configuring the parameters of the supplied insulation.

4) It is important that initially two types of isolation are loaded in the project, but you can create your own type based on them.

5) Be sure to fill in the values of the parameters: **comment to the standard size** (the parameter that will be responsible for the name), **marking of the standard size** (parameter for the V1M2B plugin), **Adsk_unit of measurement** (responsible for the units of measurement that will be included in the specification) (pis. 51).

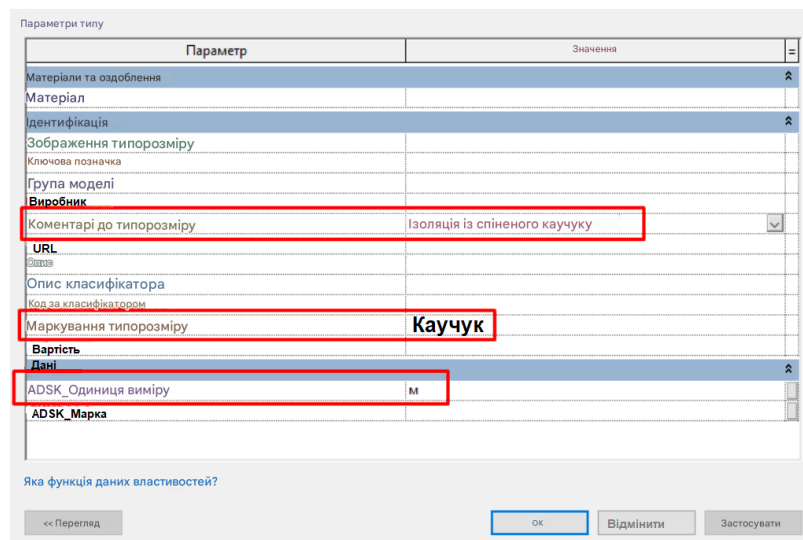


Figure 51-Filling in parameters

6) check and confirm all completed changes.

Location of insulation using specifications.

When using this method, you must configure the specification sorting correctly.

1) In the **Project Manager**, in the **Details/Specifications** category, expand the **pipelines** folder, open the O_trublines specification **O_Трубопроводи**(fig. 52).

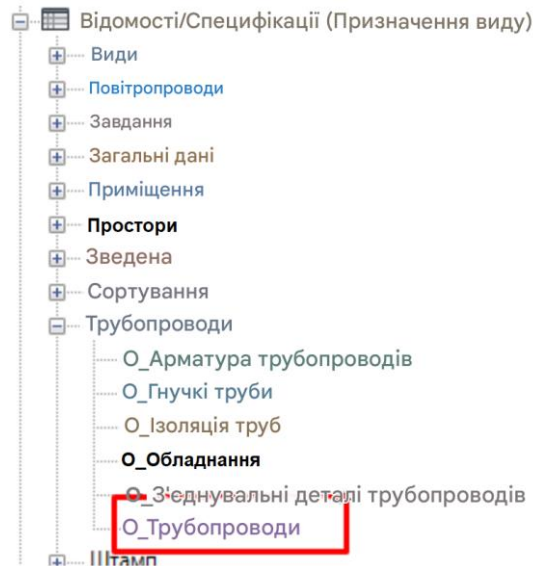


Figure 52-Selecting a specification in the project manager

2) this specification contains information about all pipelines constructed in the project (G.p53).

A	B	C	D
	Труба із захитого поліетилену 16x2,2		
	Труба мідна дюймова 15.88x0.89		
	Труба поліпропіленова PN10 #32x3		
	Труба поліпропіленова PN20 #63x10,5		
	Труба сталева водогазопровідна 15x2,8	ГОСТ 3262-75	
	Труба сталева водогазопровідна 100x4,5	ГОСТ 3262-75	

Figure 53 - Example of pipeline specification

3) when you select a row (or rows) with the required elements, the data for pipes with a specific name will be highlighted in the project.

4) activation of any plan or 3D view for access to objects. Elements of the specified type will be selected and available for further configuration (Figure. 54).

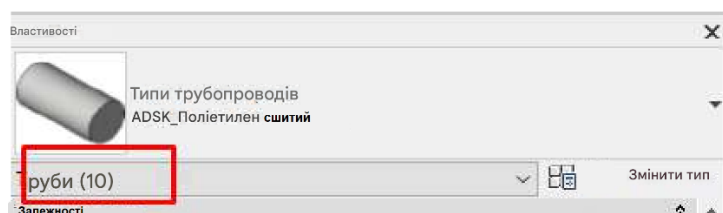


Figure 54-Properties Panel

5) adding insulation to the selected pipes according to the method described earlier.

When placing insulation, you must take into account the following features:

- Do not apply multiple types of insulation to a single pipeline. If the insulation has a multi-layered structure, you must specify this in the parameter **comments to the standard size**.

- It is not allowed to apply insulation only on a part of the pipe. Insulation is applied immediately to the entire section (from connection to connection).

- When designing, it is recommended to include isolation visibility on auxiliary plans to avoid overlapping network isolation between each other.

3.3 Basic principles of building a ventilation model

The main elements in the development of a ventilation system are: air ducts, duct fittings, equipment, duct insulation

Let's consider the features of the formation of these elements.

3.3.1 Recommendations for building air ducts.

Before you start designing the system, you need to create an auxiliary ventilation plan. The element is created using **the duct tool on the systems tab**.

The construction stages are similar to those described earlier in the pipeline system design description:

1. Define and select the system for which you plan to design.
2. Setting the level and mark values for the element (Figure 55).

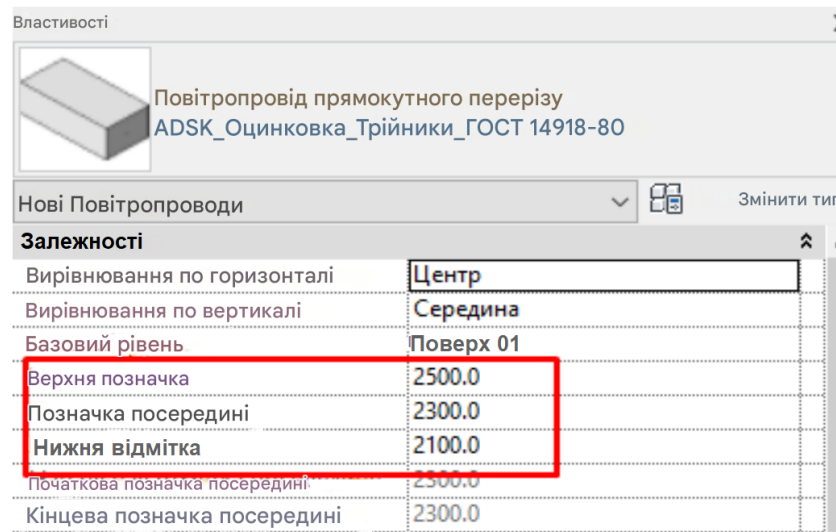


Fig. 55-Configuring the duct parameters for construction

Features of building air ducts.

During the design process, you can redefine the system name, but you must do the following:

- After hovering the cursor over a part of the system, press the Tab key several times Tabto select all the elements that make up the system (pis. 56).

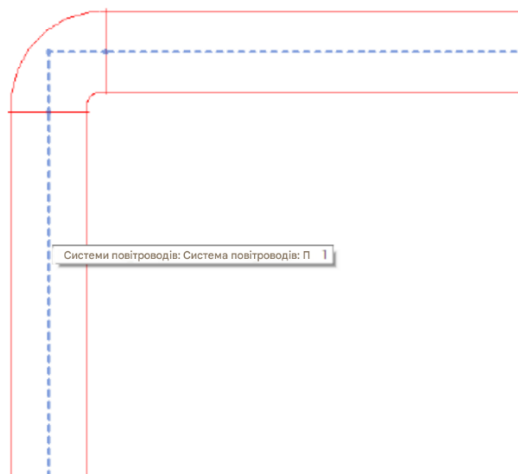


Figure 56-Selection of the built system in the design area

In the properties panel, change the system name. It is important to make sure that the entire system has the same system name and must be sealed, i.e. the connection is made only between elements of the same system. Connecting parts of air ducts are attached at almost any angle (with the exception of negative ones).

- When using Category B air ducts **B**, you must add internal insulation for them. **By default**, the thickness is 1 mm (pis. 57). This is necessary so that the thickness of the air ducts is attributed correctly.

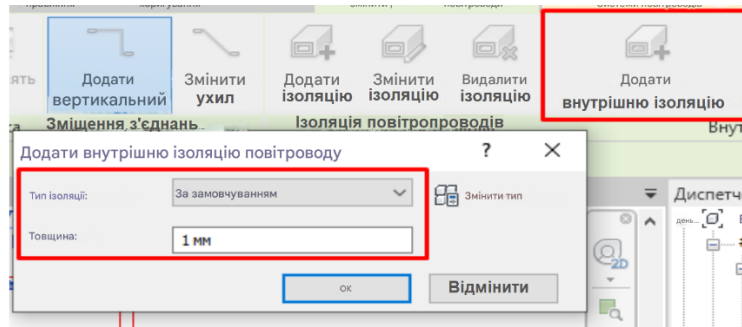


Figure 57-Adding internal duct insulation

2.3.1 Arrangement of duct fittings

The principle of placing duct fittings is similar to the method of placing pipeline fittings. We recommend that you follow the following procedure:

- 1) activate the tool on **the systems tab** (fig. 58).

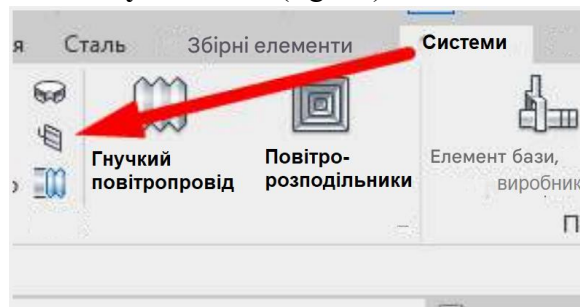


Figure 58-Rebar placement tool

- 2) Select the desired valve type (fig. 59). It is important that armature family is loaded in the start template. You can do this yourself or create a VIM Project Coordinator task.

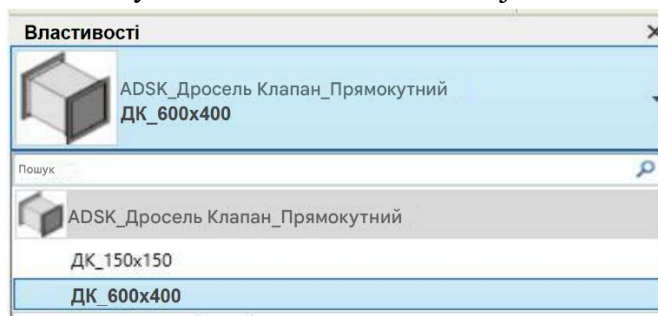


Figure 59-Valve selection

3) installation of the valve is performed by pointing at the duct axis and confirming the location by clicking the left mouse button.

Rebar placement features

When uploading ADSK fittings to the projectADSK, you need to create an additional type to configure the values of the required dimensions for this purpose:

1. Select an armature in the project and click **Change Type in the Properties panel**.
2. create a new type based on the existing one (pis. 60).



Figure 60-Creating a new type based on a copy of an existing one

3. change the size values in the new type to the corresponding ones (fig. 61).

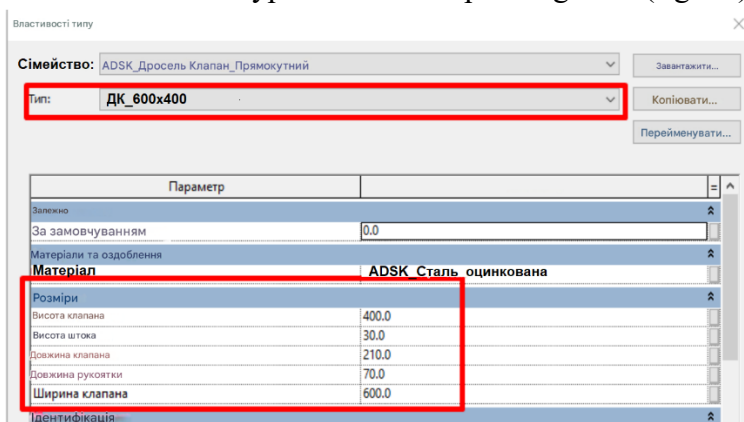


Figure 61-changing dimensions for a new type of valve

3.3.3 placement of ventilation equipment

The method of placing equipment is similar to the principles of placing equipment when designing a heating system:

- 1) select **the mechanical equipment tool on the systems tab**.

It is important to keep in mind that the starting template does not include the armature family. You can upload the necessary families yourself or send the corresponding task to the project's VIM coordinator.

- 2) select the necessary equipment for the project (fig. 62).

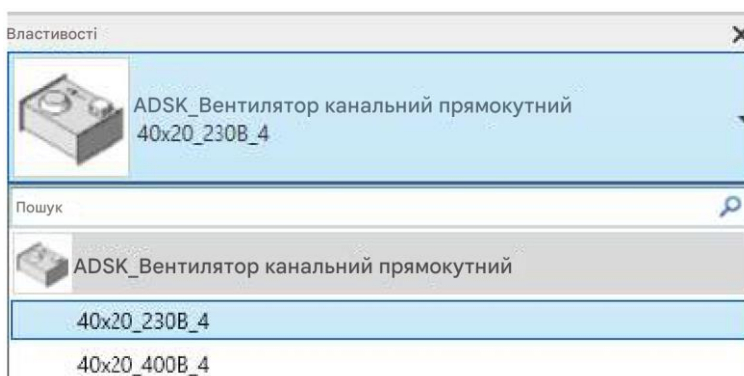


Figure 62-Fan selection

3) place the equipment for connecting networks or install it on the duct axis. In this case, the element will connect automatically.

The principle of working with ventilation equipment is similar to the method of setting up the category of equipment for the heating system.

2.3.2 Principles for configuring duct insulation

Applying insulation to the system is similar to the method discussed earlier when working with a heating system:

- 1) Select an air duct or duct network.
- 2) activate the add insulation tool (fig. 63).

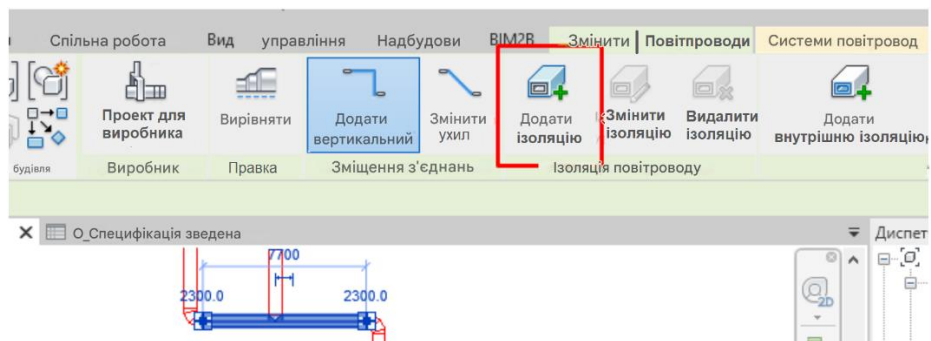


Figure 63-Selecting the Add insulation tool

3) Select the desired type of insulation (or create your own based on the existing one), set the thickness.

- 4) confirm the settings by clicking OK in the dialog box.

For this project Revit, heating and ventilation systems for the entire building were developed in the Revit environment (pis. 64).

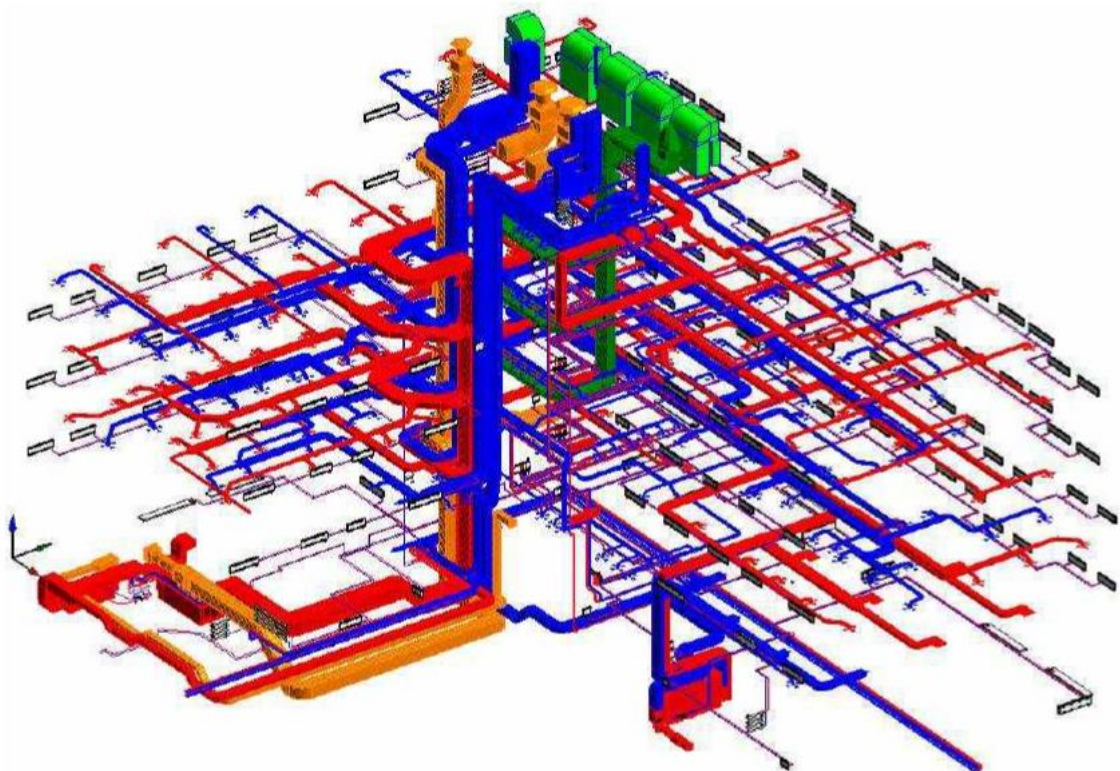
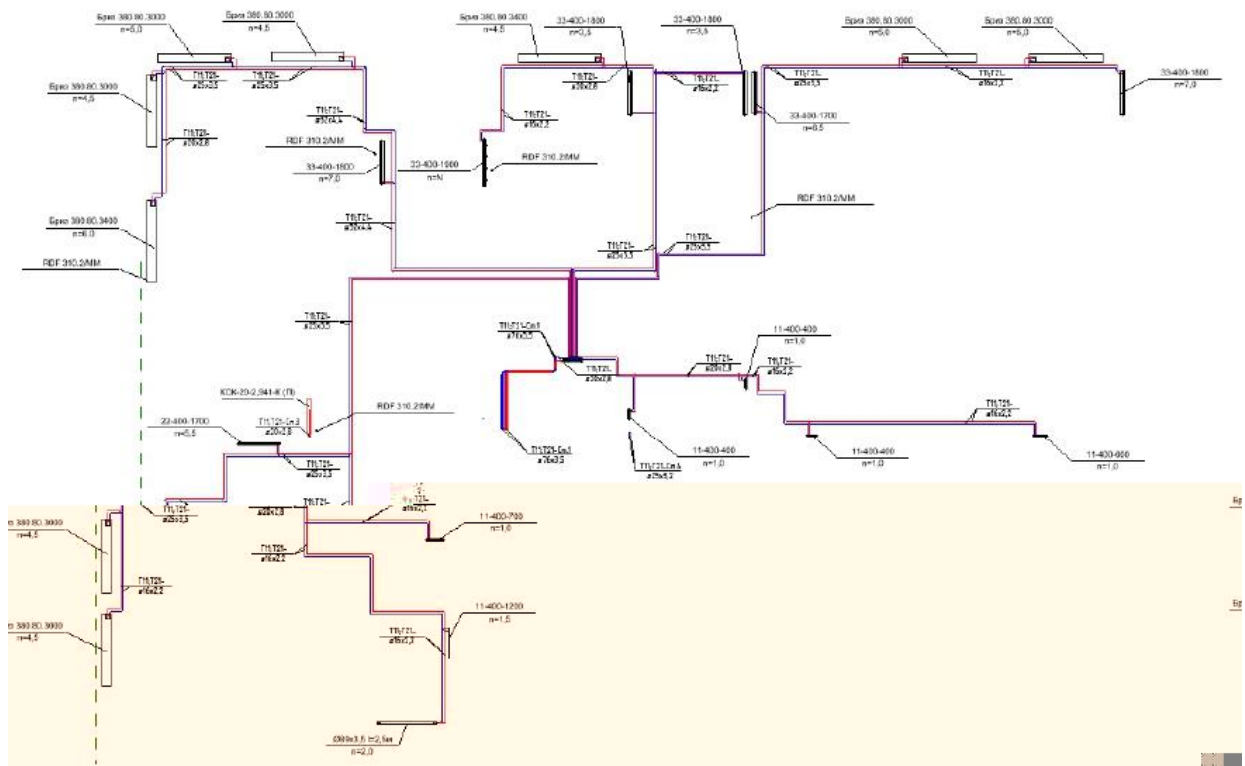


Figure 64-Designed building systems

Ha pis. 65 shows the heating system of the 1st floor. The equipment for each floor is selected, placed in the project and connected.



65-1st floor heating system

3. FILLING THE BUILDING MODEL WITH OS ENGINEERING NETWORKS IN THE LINEAR PROGRAM

When developing design and operational documentation, design companies and designers of OS engineering networks have many difficulties. There is no general approach to project development in the Bim model. Each designer works in their own calculation programs, most often these are tables in MS EXCEL, which are understandable only to themselves. When making calculations, there is no common base that you can rely on, and that will be available to all project participants, even your colleagues on the network. After performing a certain set of operations, you have to do the same steps again and again, each time checking yourself for mistakes due to inattention. Performing duplicate actions wastes the time of the designer, the company's management, and the customer.

The solution to this problem is software from the German company LINEAR [22]. This program is designed specifically for the Autodesk Revit platform for designing heating and ventilation systems. The work is performed directly in the model with full integration of OM calculations, effectively interacting with the VIM within the framework of integrated design.

Figure. 66 shows the main design steps in LINEAR software.

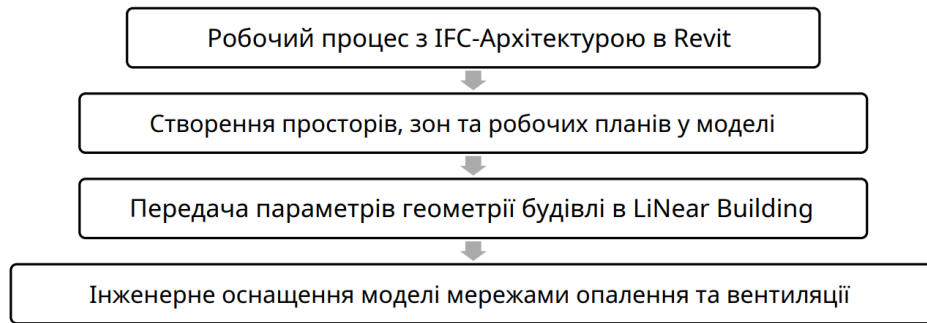


Figure 66-Design stages

Workflow with the IFC architecture in Revit

The engineering network design is always based on the building model. For a general approach, the process of working with the IFC architecture is considered. The Revit link function can also link Revitmodels, drawings from a MAP, or point clouds to the current project. The process of working with IFC-AP can be divided into several important stages (Figure 67).



Figure 67-The process of working with IFC-AP

The administrative building architectural model is used as a link.

Checking the IFC file, setting up the project

Before linking the IFC file in a new project in Revit, you should use the viewer to make sure that structural elements such as walls, windows, floors, and roof have been correctly classified, exported the author's program, and are largely free of collisions. In addition, you need spaces like IFC elements, so that you can conveniently transfer the model to Revit.

Consider the example of Autodesk Navisworks. Open it and download it from the project folder. Using **the Selection Tree structure**, you can visualize different floors (Figure 68).

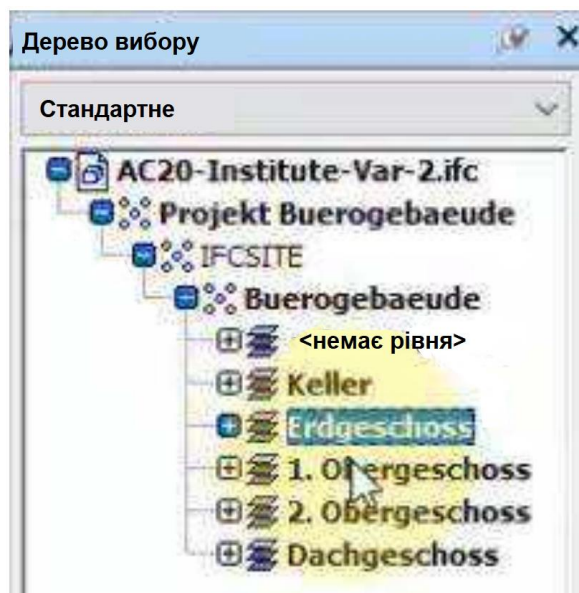


Figure 68-Selection tree in Autodesk Navisworks

For example, let's select one room and try to turn it off or on by clicking **hide**. The object's properties contain the IfcName and LongNameOverride parameters **IfcName** i **LongNameOverride**(Figure 69), which will be addressed later in this section.

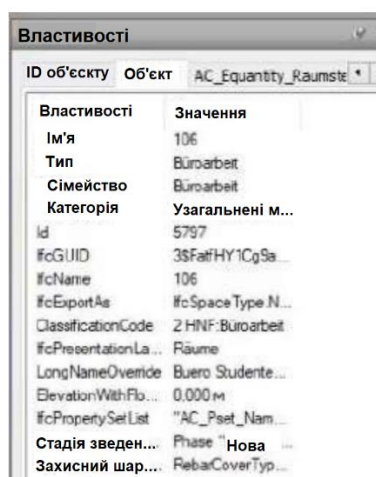


Figure 69-Space property in Navisworks

If one of these conditions is not met, you must coordinate the requirements with the architect to edit the export classification, output filter, and geometry creation characteristics. In this IFC, next to the premises, there are walls, windows, floor coverings and roofs.

Go to the launch of Linear CAD 25. If several CAD programs are installed, a selection window will appear. Choose Autodesk Revit 2025. On the left side of the projects tab, create a new project based **Шаблон on the Linear template**. Click Ok. editing starts. This template contains only the most necessary families. All necessary drawings and views are dynamically created in Linear Desktop during design.

Assigning IFC classes in the Revit category.

Before binding the IFC file, you can check whether these IFC classes and types are assigned to the corresponding Revit categories. On **the LiNear tab**, you can open and edit the IFC import configuration tables (pis. 70). The class and type can be specified at anytime.

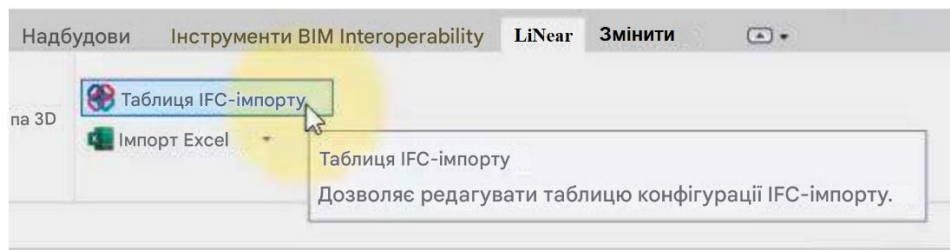


Figure 70-IFC import configuration table

An overview of the specified default types and their values is provided in the IFC specification from building smart [23]. For more information about configuring the IFC interface in Revit, see the Revitguide provided by Autodesk in English [24].

In the future, we assume that the IFC connection is performed with standard settings.

Classes, types, and disciplines of systems.

System types are used to classify pipes, ducts, and components. The system type consists of a liNear discipline, a liNear system class, and a Revit system class (pis. 71). By default, the name of the system type consists of the abbreviation of the discipline and the name of the liNear system class. System types are assigned the color and calculation type of the system. Based on the assigned system type, pipes, ducts, and elements are recognized when analyzing the pipeline and ventilation network, and their dimensions are determined.

Before designing pipelines and ducts, by selecting the liNear system discipline and class, assign the system type to automatically assign pipelines and ducts during design.

System classes. The liNear system classes and Revit system classes differ from each other. Revit system classes are clearly defined and cannot be supplemented. To use additional system types required for pipeline and ventilation network calculations, liNear system classes are required. Each class of the liNear system is a class of the Revit system and is assigned to a specific calculation type.

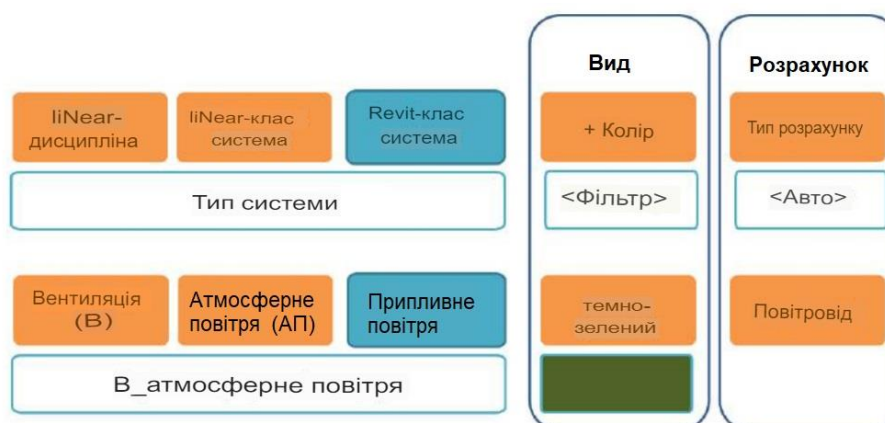


Figure 71-Concept of system types

The **System Classes dialog** shows all liNear system classes and their properties. According to this information, system types are defined. Each system class is assigned: discipline, system class, and розрахунку liNear calculation type, as well as an abbreviation and color representation of pipelines and air ducts. You can create new liNear system classes or modify existing ones.

Disciplines. Disciplines are the top organizational elements. Each element of the model is designed for any discipline.

Disciplines perform the following tasks:

- providing relevant functions on **the design tab**.
- providing the appropriate views on **the Project tab**.
- providing the appropriate system classes on **the design tab**.
- provision of relevant libraries on **the Library tab**.
- manage the display of pipes, channels, and elements on the model.
- name of system types in the design of pipelines and air ducts.

Communication with the IFC architecture

Initially, the control panel should contain standard settings, since Linear Cad was just launched. This means that **Проект** the architecture discipline is selected on the first Project tab **Архітектура**(рис. 72). By clicking **the link button**, we have the option to add AR as an IFC.

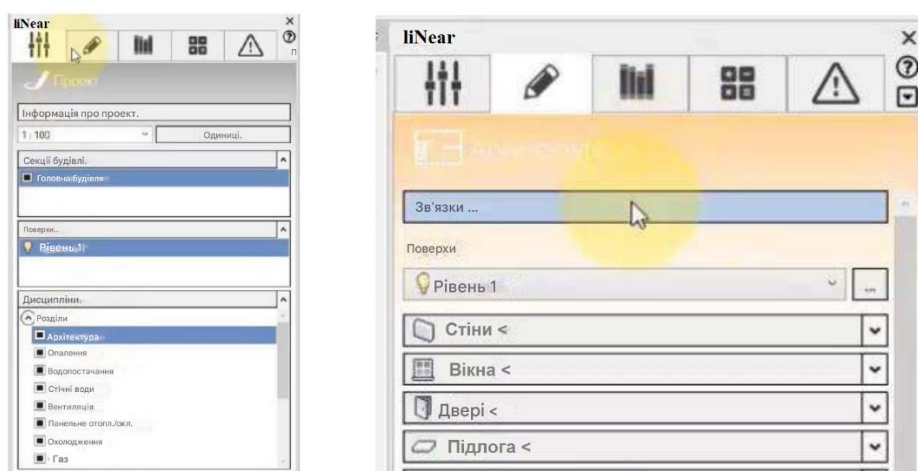


Figure 72-AR control panel

In the background, Revit automatically creates the .ifc, rvt extension file. This file contains the Revit model, which is a representation embedded in the IFC geometry and an attribute. It should be noted that the generated model can be opened and partially modified in Revit, but full editing is not possible. Elements such as walls, windows, ceilings, and doors were correctly categorized in this model, but editing using the usual Revit tools is not possible. This way, a link to the AP will be created.

Import floors. We will import the levels included in the IFC model into our model with engineering systems, and delete the level that already exists in it. A project should have only one level system.

Under **the link button**, you can see the floor selected for building and a table with data by floor. In this dialog box, you can import and synchronize levels with IFC AP. After creating the levels, create the corresponding views.

Ability to visualize, disable marking levels from links

The view may also show link levels that contain height data in meters (рис. 73), although values for other levels are displayed in mm.

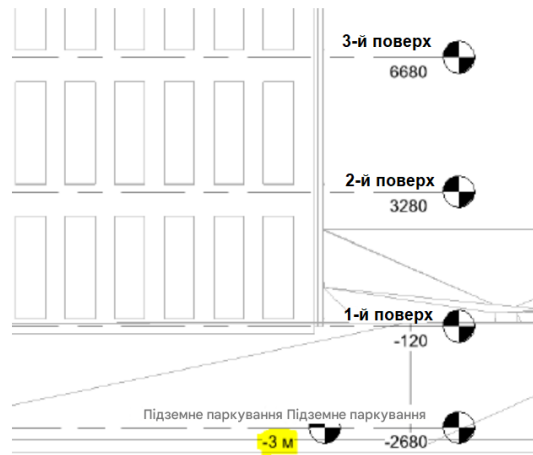


Figure 73-Equal models

You can disable link labeling in **the Properties tab- redefining the visibility of graphics** for this view (pis. 74). After you disable the visibility of levels in the linked AP file, only the levels of the current file are visible.

To improve the presentation, we recommend disabling generalized models from the link. This usually means that graphical representations of the room, such as openings, are no longer displayed so clearly.

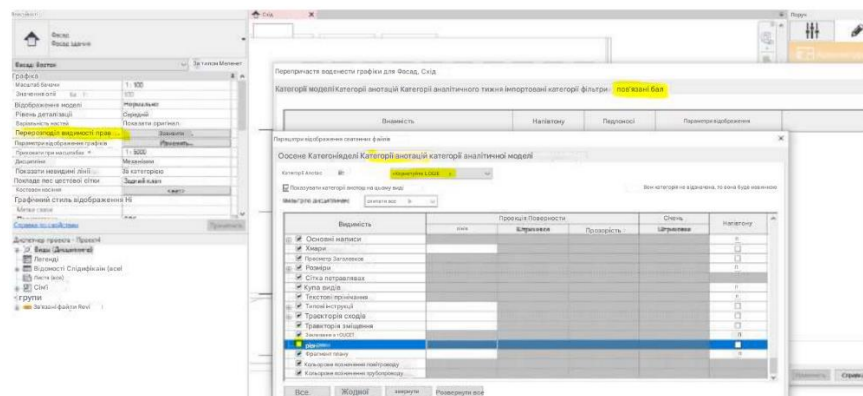


Figure 74-Configuring the appearance

For demonstration purposes, you can select fixed elements on this view by unpinning the pin icon at the bottom right of the workspace. Now, by pressing the tab key, elements from the link are selected. Thus, we can see that the protective layer of reinforcement, a generalized model, covers the windows. In the Disciplines section of **the Architecture category, the Model Body** in this form parameter is disabled, including the link (pis. 75). This way, you can easily detect errors in the IFC classification table.

In case of errors, the binding procedure will generate generalized models instead of the correct categories, as a temporary solution. This can lead to technical problems in the following processes: calculation of heating and cooling loads, design of openings and openings.

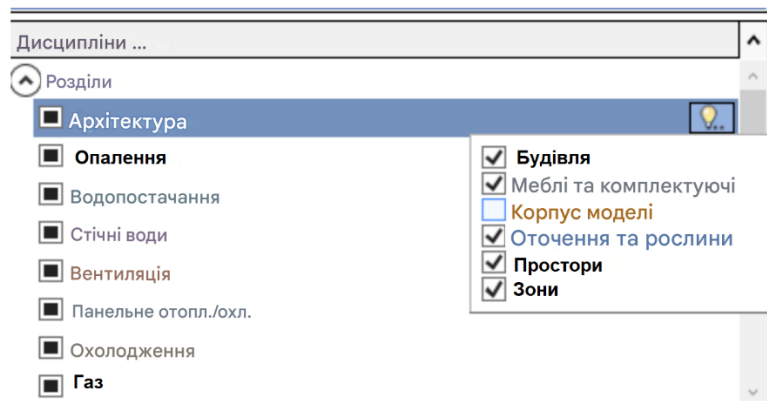


Figure 75-Editing the display to detect IFC AR errors

Setting the calculated height. By clicking on any level marking, you can set the calculated height for spaces in the properties window.

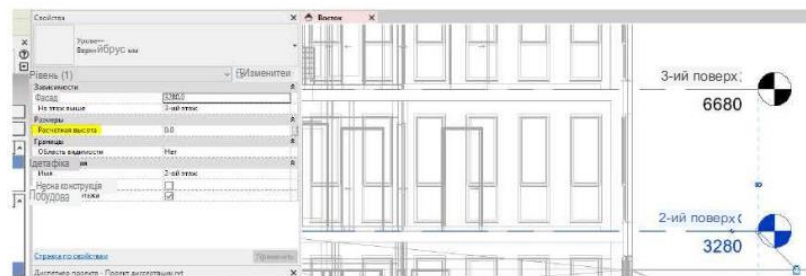


Figure 76 - Setting the calculated height of the space

You need to create a plan view for each floor to place spaces on them. To do this, in the view management area of the control panel, select the appropriate plans icon and click on the green plus sign to create new plans (Figure 77). In the window that opens, all the options will already be set by default.

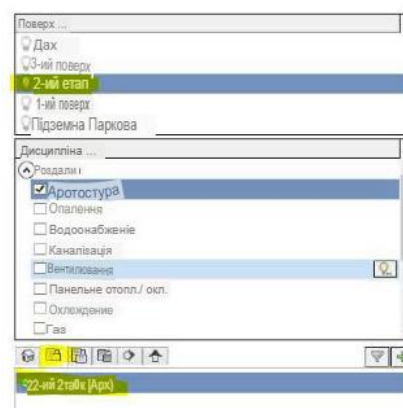


Figure 77 - Creating plans

When switching between floors, the corresponding view is always suggested in view management. The name of the plan is set as the embedded schema configuration. Any inactive view can be renamed, named by default, or deleted at any time.

On each type of main level, you can create the spaces necessary to determine the heating load.

Premises in IFC are created as generalized models and are therefore selfless for further engineering system design. To be able to calculate the load in the future, you first need to create the appropriate space, as in the Close BIM workflow.

To create spaces, go to the **architecture discipline**, on the structures tab on the control panel. The creation of spaces is carried out floor by floor, on the corresponding plan. Under the bundle tab, you can select a floor. When you change a floor, the corresponding floor plan is automatically generated in this window.

Before creating spaces, you need to check the data transfer settings from the architecture. At a minimum, room numbers and names must be imported. By default, the \$Name and \$Number variables are used for room names and numbers \$ Number (Figure 78).

Using **the apply additional parameters button**, you can, for example, add apartment names if this parameter is already defined. Thus, apartment-by-apartment division can be implemented in LiNear Building.

We see that the lists display the names ifcName and LongNameOverride, which were previously visible in Navisworks (Figure 79). You can use them to create numbers and names of rooms, so that this data can be transmitted already during the creation of spaces.

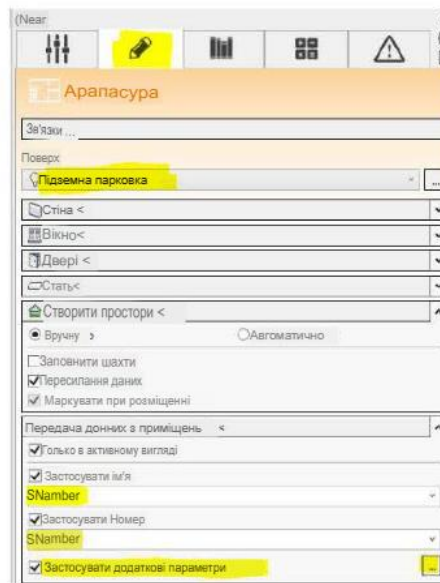


Figure 78-Accepting room data from a linked file

Creating spaces let's start with the basement, using the mayor's creation of spaces, adding them manually or automatically.

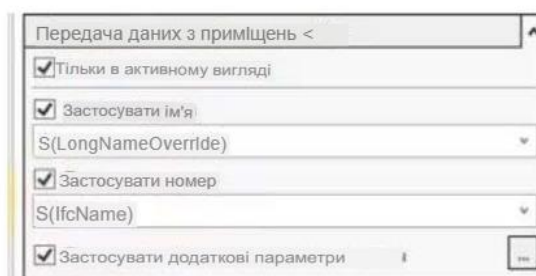


Figure 79-Additional parameters for transmitting data from premises

To automatically create spaces, you can activate automatic labeling. The automatic creation method saves a temporary resource, since it requires only two mouse clicks for each level and a full transfer of room data in the ready space.

At the end of the creation, a message appears in the lower-right corner indicating howmany ib and mine spaces have been created (Figure 80).

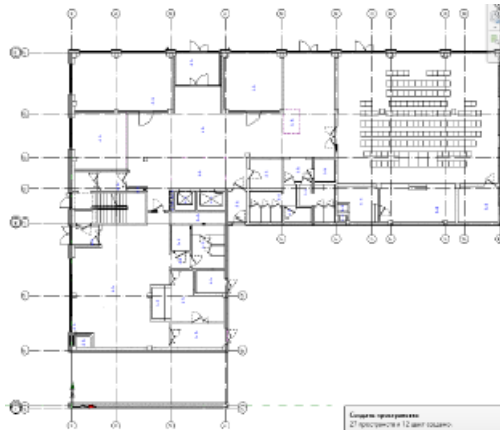


Figure 80-1st floor plan with notification of created spaces and shafts

On the first floor of the building there are 27 rooms, 12 mines. Spaces on floors are defined according to the specified height of the rooms from the top to the upper boundary of the next floor. Spaces on other floors are created in the same way. By selecting one of the rooms, you can control the upper limit and offset. In each case, the upper limit should be set to the next floor without any offset. This elimination inand is shown in **the property** (Figure 81).



Figure 81-Space properties

На останньому поверсі There are technical rooms on the top floor-there is no next floor, so the offset equal to the floor height is indicated. prostir finds the roof structure at all points.

Image of spaces. Each space is a zoning element. Allcreated places створюються are initially located in **the default zone**. To divide spaces into different zones, it is useful to visualize them to make it easier to navigate the building. The space can be displayed on any of the existing views by selecting **the show function**.

Using the 3D Selection Frame toolD, you can view any room or space separately in 3 dimensions (Figure 82).



Figure 82 – 382-3 D Separate space view and location in the model

Zoning system. Zoning allows you to centrally adjust the desired temperatures for heating and cooling modes. select, rename them, or assign the appropriate temperature settings.

For areas of technical rooms, stairwells, attics, and basements (Figure 83), you must set the air temperature value according to the standards [25].

In the **System Types** pane, you can specify a name for the pipeline system. You can choose either an abbreviation or a fully spelled-out name to indicate a discipline.

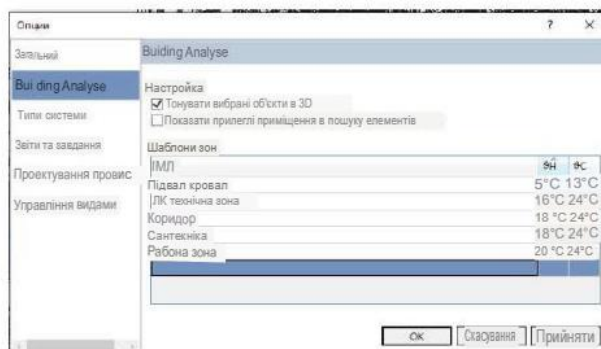


Figure 83 - distribution of premises by zones

Individual zones and temperature regions can be displayed by creating a 3-dimensional display (Figure 84). The level and zone buttons allow you to display two options for assigning spaces. It is allowed to change the value of heating temperatures in the zone properties. After adjustment, the data in the engineering systems file will be updated. If the spaces no longer correspond to one zone, you can determine whether they belong to another zone. As a result, the model is ready for data transfer.

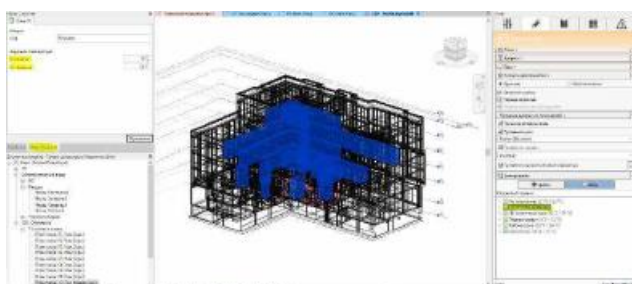


Figure 84 - Selected corridor area in the model

The section on creating workspaces, zones, and plans for calculation allows you to comprehensively analyze the building model, see the future work front, and reset the ballast from transmitting model geometry parameters.

Transmitting parameters and analyzing the building's geometry

You must activate **the Analysis tab**. If you start transmitting data to LiNear Building immediately, the properties of materials from multi-layer walls will not be transmitted to the IFC model. If elements such as walls, windows, and doors already have heat transfer coefficients, then the corresponding parameter can be set by the configuration. However, you should make sure that reasonable values are set at this transmission point. If you are not sure about this, you should not trust the specified values. Instead, simply set the layer structure or heat transfer coefficients in the main project file table in linear Building.

In this case, there are no heat transfer coefficients with the SHS. To understand where these parameters should be specified, open **the parameter management tab**. In this dialog box, you can set several parameter aliases, and there are differences on the screen for different elements (Figure 85).

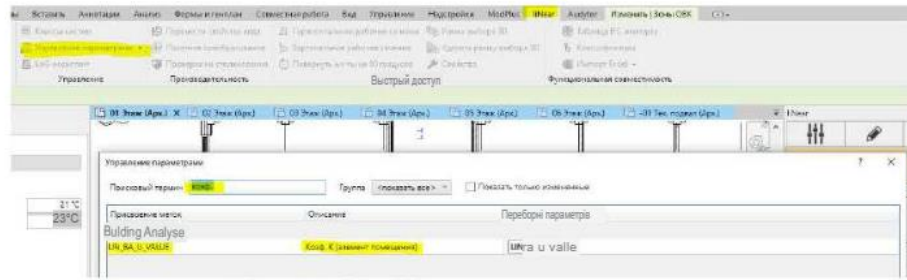


Figure 85-Parameter management

Open Building by clicking **Analyze Building Data** (Figure 86).

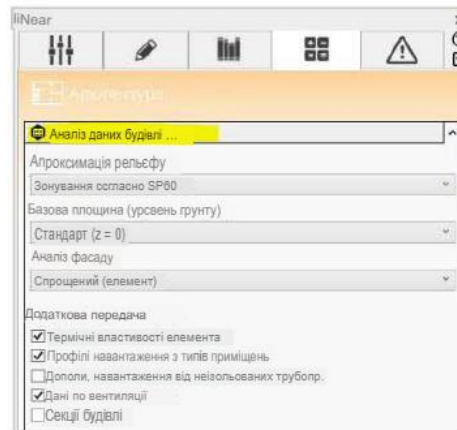


Figure 86-Starting Building Data Analysis

Let's analyze the building data based on a standard template (Figure 87).

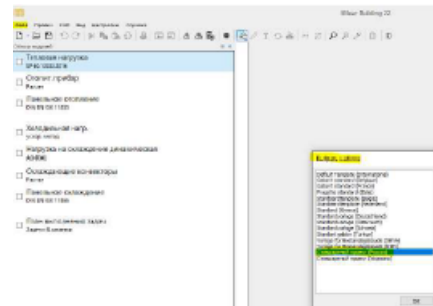


Figure 87-Template for calculating the thermal load of a building

A message about available reports appears (Figure 88). Then a dialog box opens for entering general building data.

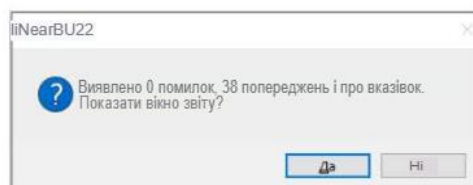


Figure 88-Report window with building design warnings

Select the type of building (in this case, an office / administrative building) with a foundation slab depth of 3.2 m (Figure 89).

Дані будівлі

Будівля

Тип будівлі:

Зона вентиляції:

Проект:

Одиниця будівлі:

Враховувати квартири з однаковою назвою і номером як одну квартиру

Геометрія

Довжина будівлі:

Ширина будівлі:

Площа:

Висота будівлі:

Об'єм будівлі:

Пол всіх опалювальних приміщень, що стикється з ґрунтом

Довжина:

Ширина:

Площа:

Периметр:

Глибина до ґрунт. вода:

Глибина фундам. плита:

F9-взяті розміри з CAD

Figure 89 - Entering data about the IVLC

Then we will go to the **thermal load module**, the parameters of which correspond to DBN B. 2.5-67: 2013 "Heating, ventilation and air conditioning" [26].

Загальні дані | Теплове навантаження SP | Перелік матеріалів

Загальні дані по опалювальній навантаженні

Місцевість:

Темп. зовніш. повітря: °C

Поправка: K

Расч. сварок, вітру: м/с

Тип місцевості:

Вологість: суха норм. овлажнена

Не враховувати втрати тепла через внутрішні конструкції при різниці температур в гомеостаз 3 ° C і менше

Будівля

Внутрішній тиск: Па 7 Па

Спрощений розрахунок інфільтрації

Т-ра прит. повітря: °C

Висота будівлі до витяжної шахти: м

Довжина будівлі: м

Ширина будівлі: м

Площа підлоги АПп: м²

Q побут: Вт / м²

Типовий проект

Коеф. псправки

B1:

B2:

Figure 90-Input of general heating load data

The report is examined in Revit. The error about "overlapping elements" is standard and appears frequently (Figure 91) and reports overlaps overlapping each other. It was created due to the construction of the building itself, which overlapped with the IFC model. You can generate correction tasks from reports, which you can add comments and screenshots to.

On the **topics tab**, you can create a group and share it with all project participants. Editing reports and tasks can be documented.

The zoom command in the LiNear Building control panel turns out to be an element that belongs to the selected line, including in related models.

The **Eyedropper search** tool helps you find items that are selected in Revit that match table rows. For these commands, you can use a 3D floor view with semi-transparent tinting, where the **generalized model category is disabled**.

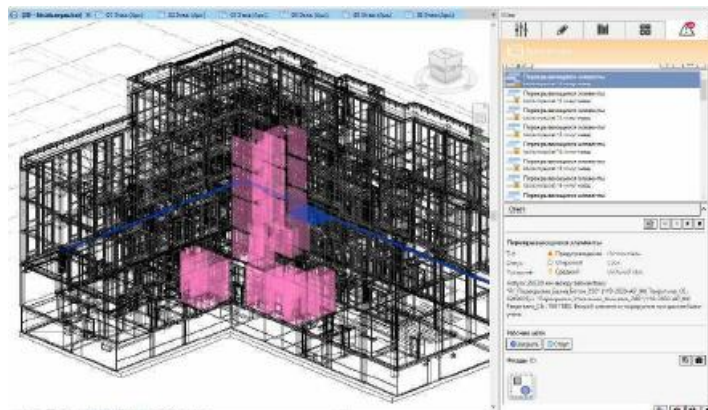


Figure 91-A common misconception about overlapping AR elements

The structure of the building is correct, but messages about heat transfer screening factors appear in the heat load module. You can enter them in the main file table. For heating loads, all heat transfer coefficients, except for elements in contact with the ground, can be set. To calculate the dynamic cooling load, a detailed layer structure is required

3.1 equipping the model with a heating system

This section covers the main processes of working on the development of a heating system:

- Thermal characteristics of the building
- Calculation of building heat loss.
- Calculation and selection of heating devices.
- Transfer of devices to the project.
- Heating of the premises with underfloor heating system.
- Tracing and calculation of the heating system.
- Placement of the heat generator.
- Cross-section traversal.

Thermal characteristics of the building. Calculation of heat losses of houses will be performed in the LiNear Building program in **the Building structure module**. Rooms that are deactivated for calculating the heating load are displayed in the building structure with the symbol-a red circle with a white minus sign. Such a room was taken into account in the analysis, but only the temperature is transmitted to the adjacent rooms.

The red triangle symbol with an exclamation mark indicates a warning, which is described in more detail in the protocol window in the lower rightcorner of the program window (Figure 92).



Figure 92-marking of indoor errors

To the right, next to the building structure, the general parameters for calculating the heating load are shown first.

Error analysis. By clicking on the notification button on the left side of the protocol window, you can see a warning for the activated module. Consider one of the messages by clicking on it (Figure 93). Go to the element where the error occurred.

You should carefully review this message to make adjustments. In the case of an unspecified heat transfer coefficient for the window: $K=0.00$.

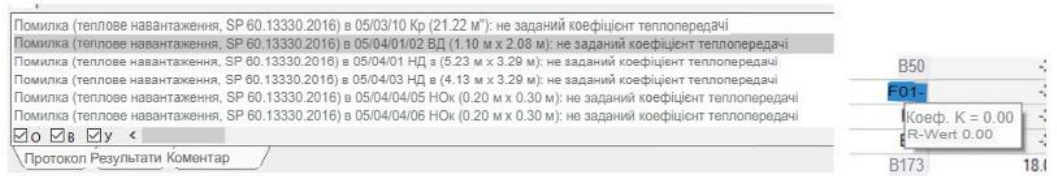


Figure 94-error about no heat transfer coefficient

Determines the heat transfer coefficient of the window. As a rule, it is located in the list of recent messages. After selecting the list, the program will go to the main tables of the file. The name of variables depends on the order in which elements are created on the model, and therefore the heat transfer coefficients of elements, even if they have the same designation, may differ for each project.

The main tables of the file contain a search variable for the Revit element area. There are even more errors in the window and door areas, and even the available heat transfer coefficients do not meet the requirements.

Dormer windows with a heat transfer coefficient of 5.560 W / m² K are not relevant. If the corresponding norms of heat transfer coefficients differ from those specified in Revit, then we change the values (Figure 95).

№	Тип	Назва	Коефіцієнт
1	Двері	П_Двері_в_коридор_01_01	1.000
2	Двері	П_Двері_в_коридор_01_02	1.000
3	Двері	П_Двері_в_коридор_01_03	1.000
4	Двері	П_Двері_в_коридор_01_04	1.000
5	Двері	П_Двері_в_коридор_01_05	1.000
6	Двері	П_Двері_в_коридор_01_06	1.000
7	Двері	П_Двері_в_коридор_01_07	1.000
8	Двері	П_Двері_в_коридор_01_08	1.000
9	Двері	П_Двері_в_коридор_01_09	1.000
10	Двері	П_Двері_в_коридор_01_10	1.000

Figure 95 - Window with element coefficients

The same procedure applies to doors. The heat transfer coefficient is not available only for one door, but also for other doors, the assigned values correspond to the current requirements (Figure 96). Similar actions are performed for walls and ceilings.

№	Тип	Назва	Коефіцієнт
1	Двері	П_Двері_в_коридор_01_01	1.000
2	Двері	П_Двері_в_коридор_01_02	1.000
3	Двері	П_Двері_в_коридор_01_03	1.000
4	Двері	П_Двері_в_коридор_01_04	1.000
5	Двері	П_Двері_в_коридор_01_05	1.000
6	Двері	П_Двері_в_коридор_01_06	1.000
7	Двері	П_Двері_в_коридор_01_07	1.000
8	Двері	П_Двері_в_коридор_01_08	1.000
9	Двері	П_Двері_в_коридор_01_09	1.000
10	Двері	П_Двері_в_коридор_01_10	1.000
11	Двері	П_Двері_в_коридор_01_11	1.000
12	Двері	П_Двері_в_коридор_01_12	1.000
13	Двері	П_Двері_в_коридор_01_13	1.000
14	Двері	П_Двері_в_коридор_01_14	1.000
15	Двері	П_Двері_в_коридор_01_15	1.000
16	Двері	П_Двері_в_коридор_01_16	1.000
17	Двері	П_Двері_в_коридор_01_17	1.000
18	Двері	П_Двері_в_коридор_01_18	1.000
19	Двері	П_Двері_в_коридор_01_19	1.000
20	Двері	П_Двері_в_коридор_01_20	1.000

Figure 96-Heat transfer coefficients for doors

Only one unknown element type remains. Such elements usually occur as a result of sloppy work with the AP model.

To find out where such elements are located in the model, try removing the heat transfer coefficient. You can only remove it if it is not used. Otherwise, you will see a hint where you can define it, and the message contains the floor, room, and line. When you delete an item, a warning dialog box appears (see Figure 97).

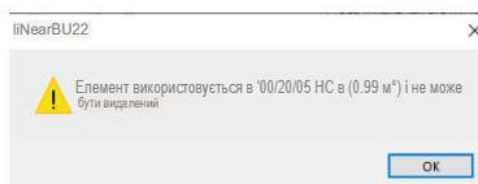


Figure 97-Delete unknown element dialog box

In this case, the element is located on the first floor. Instead of the wall, a separator line was used. Since this area is not calculated, but receives all the heat from neighboring rooms, you can ignore this warning in this example. the warning will be delayed forever.

Because even when the building data is updated, the edited assigned heat transfer coefficients are not overwritten. All necessary changes have been completed in the main tables of the file. It is important to clear the check box for transmitting thermal properties of elements on **the Building data analysis tab**, so that changes are not overwritten when building data is updated (Figure 98).

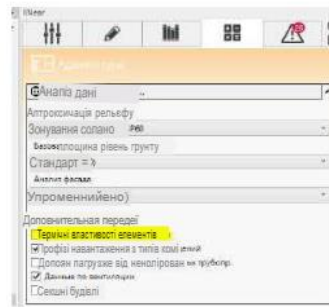


Figure 98-Configuration parameters for data analysis

In the list of building structure elements, you need to deactivate individual rooms, such as the crossover and electrical switchboard, where electric heating will be installed, and the basement corridor area by unchecking the box in the header (Figure 99).

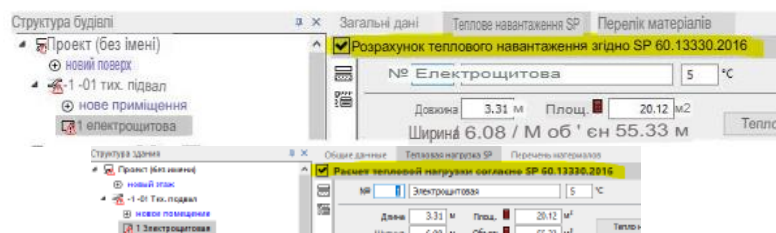


Figure 99-rooms not heated by water heating

Features of calculating the heat loss of a building according to GOS V. 2. 5-67: 2013 [26].

According to the standard, zoning is required for all walls and floors that come into contact with the ground. The components in contact with the soil are divided into zones I-IV with an interval of two meters (Figure 100).

Zone I is adjacent to the Earth's surface, while zone IV is the innermost area that extends along the components in contact with the soil, at a distance of more than six meters from their edge. As the zone number increases, a stronger insulation effect is applied to account for the uneven temperature distribution in the ground. If there is a layer structure for a component in contact with the soil, then the value is determined by the zone and the lambda value of the insulation material in accordance with the standard.

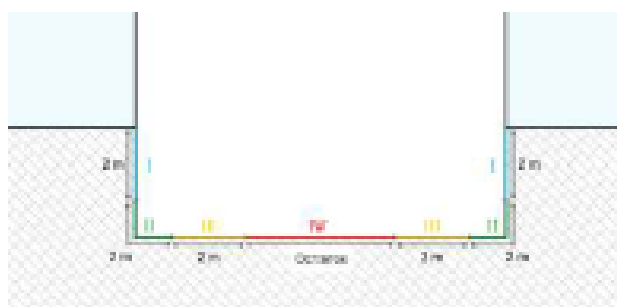


Figure 100-Load zoning scheme

Based on the defined value of R and the component's heat transfer coefficient, the total heat transfer coefficient K(o) is calculated. If the structure of the layers of the component in contact with the ground is unknown, a warning will appear in the protocol and a numeric value will be used. This warning can be deactivated in the configuration on **the SP heat load tab**.

By calculating transmission losses, the program automatically determines the heat requirements for heating the infiltration air, as well as the ventilation holes of the outdoor air (Fig.101). The largest of these values is taken into account in the calculated losses of the premises.

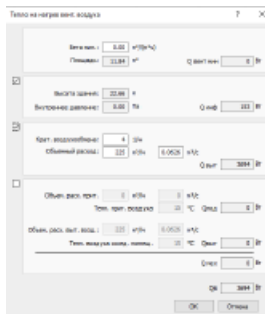


Figure 101-value of the parameters of the load on the infiltration and vent. expenses

Go to the building structure in Linear Building at the project level to see **the results window for total heat load**. The total is 113,386 watts (Figure 102).

Проект: Проект (Розрахунок теплового навантаження)	
Qогр, Вт-Трансмісійні втрати тепла	= 113386 Вт
Qвент / інф, Вт - велика з потреб в теплоті на нагрівання вент. Qвент або інфіль ... =	57582 Вт
Q-доб, Вт-додаткова витрата теплоти	= 0 Вт
Q побут, Вт-тепловий потік від побутових джерел теплоти	= 0 Вт
Навантаження з коефіцієнтами поправки (B1, B2)	= 0 Вт
Розрахункова опалювальна навантаження	= 171055 Вт

Figure 102-summary of calculations for heating equipment building load

As a result, the values obtained during manual calculation and LiNear Building are compared. The margin of error was 11% lower. The manual calculation resulted in a slightly larger load. According to the standards, an error of 5% is allowed.

You can accept the received data for project documentation, since the calculation of the building took approximately 60 minutes of working time, from the beginning of loading the model to the end of the calculation. Manual calculation with entering all data about the building, measuring each room and setting up, takes about 3-5 business days, depending on the building. This includes time to check for a typo based on human factors, which is excluded in the LiNear app.

3.1.1 Calculation and selection of heating devices

Both **Heat load modules** and **heating devices** must be started, otherwise the calculation of heating devices will not be shown.

You need to make a preliminary selection of equipment. Office premises should be equipped with flat radiators. From the list of manufacturers' technical data, select the Kermi model range of the Profil-K device (Figure 103). This radiator has certain options, which allows you to choose the connection methods. Additional components may also be available for issuing specifications

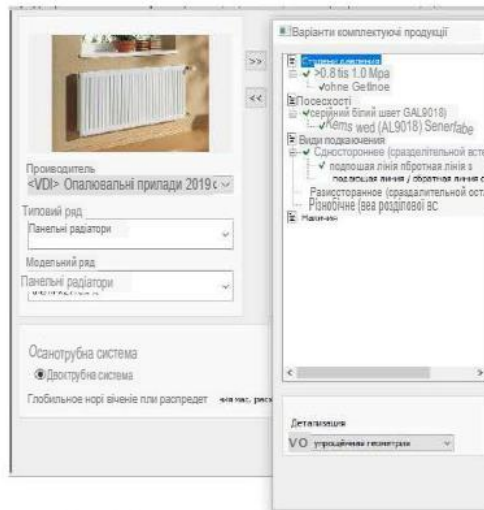


Figure 103-Radiator selection for the heating system

In the **Calculation Options** tab, you need to perform a preliminary selection of heating devices, which can be adjusted on a case-by-case basis. In this dialog box, you can limit the height, design length, and instrument type (see Figure 104).

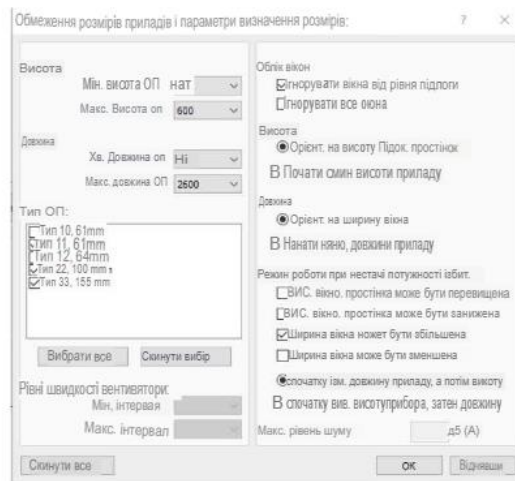


Figure 104 - Setting the limiting characteristics for the radiator

As a rule, a radiator is selected for each window according to the formula: radiator length = (Window width-250 mm minimum margin in length)

In the connection kit, you can select additional values for pipeline systems of specific manufacturers. The connection position setting is clearly shown in Figure 105.

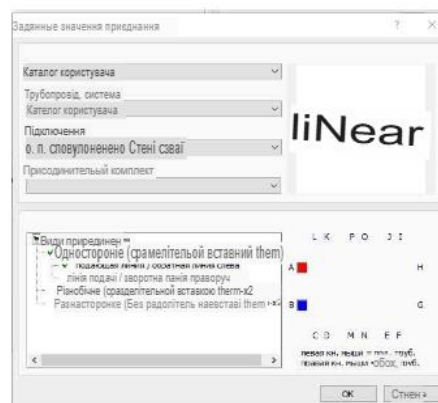


Figure 105 - Setting radiator connection values

You must set the calculation goal. It is possible that either the heat output or the temperature difference is recorded. Set calculation values are provided for each specific boiler. The values of the heat carrier from the ETC connected to the building are added. The temperature schedule for heating the building administration is 75/50 degrees.

The minimum mass flow rate should not be lower than 10 kg / h, 200 kg / h is the upper limit from which noise is expected to appear in the system. Single-pipe and double-pipe systems are also available for calculation.

The Project overview and Dataset import tabs are actively used. Using the import option, you can download any sets of technical data in accordance with international standards, that is, the design allows the use of heating devices from any manufacturer, which provides the corresponding list of data.

Вкладка **The Project Overview tab** allows you to see all the selected heating devices in a general view, and if you want to replace some of them.

The radiator was selected in the office on the 2nd floor B. Based on the heat loss in the building, the program will suggest the number of necessary radiators. Radiators are planned to be placed under the windows. For a medical center with one window, the program selected one radiator of type 33,400 mm high, 1800 mm long (Fig.

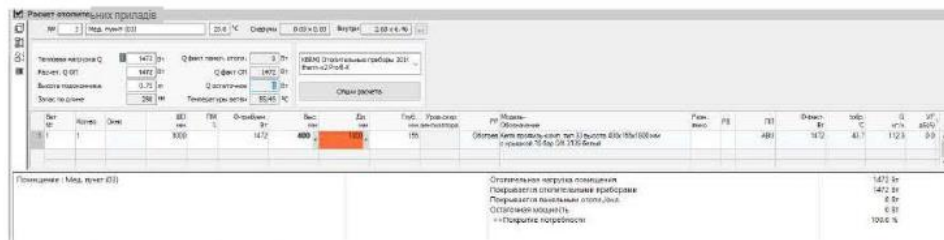


Figure 106 - selected radiators for the medical center premises

Similarly, we select devices in other rooms, as a result, we selected 3 radiators, and for another room – two. A warning message about the heating load appears in each room, indicating whether the limit has been reached or not. You can also set the limits of warnings in the configuration about failure to reach or increase the heating load coverage. When these borders are expanded, the red marking of lines containing errors disappears, but the protocol will contain warnings about possible consequences. In the LiNear Building configuration, in the radiators area, you can set the mounting height above the basement and the distance from the wall (Figure 107).

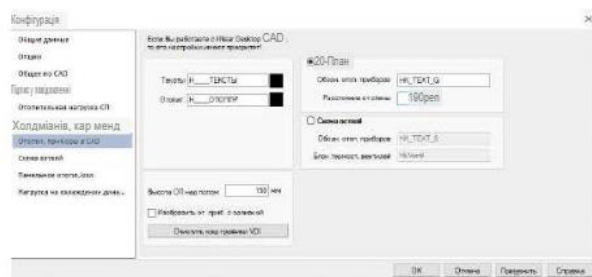


Figure 107-configuration of heating devices

After that, the selected heating devices can be transferred to Revit.

3.1.2 Adding radiators and transferring data to the project

You need to turn on the display of **heating discipline** and floor plan mark the first floor of the building and use the pencil symbol and the F10 key to start the construction process of radiators (Figure 108) as a result, they will be automatically located under the corresponding window.

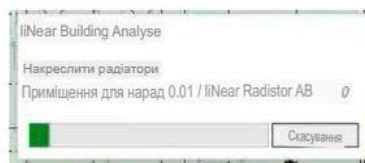


Figure 108-Automatic placement of selected radiators

The design lengths of some radiators may be indicated in red, which means that the niche length has been exceeded. Since the radiators in this project are not located in a niche, you can not take any actions to adjust. For radiators, you can change the connection side, this is indicated in the PP column (connection position).

The dialog box displays a dialog for changing the attachment position, and the data will be transferred to Revit. You can use **the update function**. Thus, the radiators are edited and plotted on the drawing (Figure 109).



Figure 109-Automatically placed indoor radiators

Heating on the ground floor of outdoor office premises is provided by indoor convectors from the manufacturer Arbonia. All rooms, including corridors, are deactivated for calculation and selection. An indoor floor convector is always added centered under the window when placed automatically and cannot exceed certain building dimensions. An installation channel with a width of 250 mm and a depth of 120 mm is installed in the screed around the building. The convector is connected to the left or right side of the room. Its construction height must exceed 110 mm. As the type was set by KKR81 210 mm, the construction length must exceed the width of the window. The selection will start with the minimum height and length.

In the configuration, the height from the floor should be set to 0 mm and the distance from the wall to 20 mm. Selection is carried out floor by floor. Let's start with a room on the 1st floor. The program offers to install 7 convectors with a length of 2000 m. After automatic placement, check the positions and heights of the convectors. By creating a horizontal section for the room, you can see that the convector is exactly at the height of the first floor. The remaining devices are selected and stored in the same way (Figure 110).



Figure 110-Placement of indoor convectors

3.1.3 Heating underfloor heating

Calculated heating for the lobby and cafe on the 1st floor.

In the LiNear Building program, open **the panel heating module**. At the project level, you need to make a preliminary selection of systems. How the system is chosen by the manufacturer Rehaus 2024, the Noppenplatte Varionova system Noppenplatte Varionova.

This system has certain parameters, the most important of which are the pressure loss and the installation pitch, according to the manufacturer, 300 mm can not be exceeded. When calculating heating circuits, the program tries to minimize the length of pipelines. specification of materials (Figure 111).

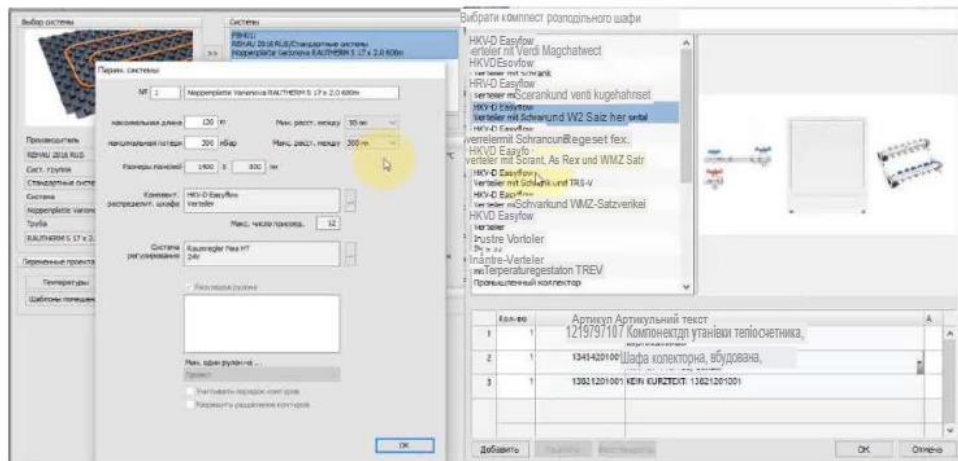


Figure 111-selection of manufacturers and materials for underfloor heating

Business along with the choice of system, you can choose additives in the screed. They are usually designed to cure and minimize the thickness of the screed layer. In the total preloads, you must specify the calculated values: feed temperature, temperature differences, and maximum floor temperature. Along with the field with the temperature of the supply pipeline, there is a calculator symbol that helps with the calculation. You can determine the required temperature in the supply line. The maximum surface temperatures are always exceeded and observed (Figure 112).

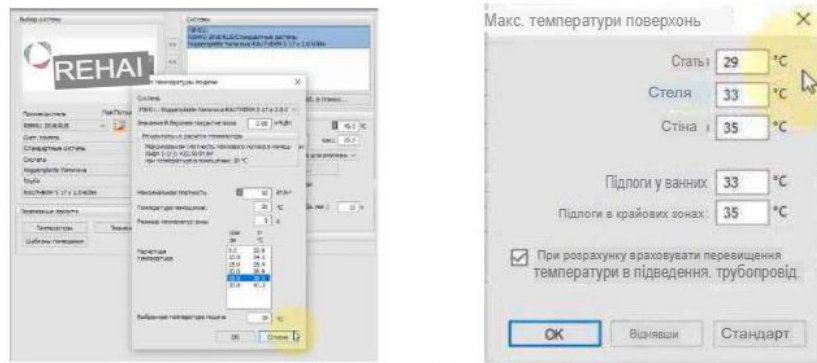


Figure 112-parameters for calculating underfloor heating

If the pipes connected to the distributor do not need to be reflected, the average connection length on the distributors for the supply and return pipes of 10 m can be indicated. This reduces the maximum contour length set by the manufacturer. After the correct system selection, all systems in the building structure are deactivated, except for the selected premises.

At the time of activation, the net heating load is indicated. By selecting one of the suggested surface materials, but this list can be edited and expanded in the main tables of the file (Figure 113). In the insulation column, select a specific type of insulation. Manufacturers have provided different insulation thicknesses.

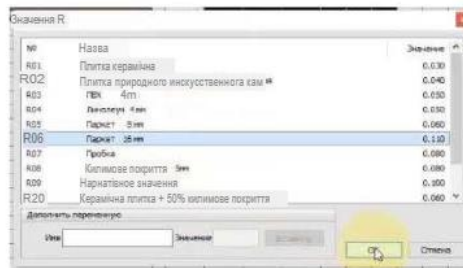


Figure 113 - inibir of floor composition for calculating underfloor heating

By clicking paintwork in the installation area line, you can add limit and edge zones, as well as blind areas and room elements (in this example, the floor).

When you select a work area, 4 heating circuits will be automatically created. The load of 3714 W will be evenly distributed, and the installation step, temperature difference, pressure loss, and pipeline length will be determined for each of them (Figure 114). It should be taken into account that with a capacity in the middle, the heating system has pressure losses to the outside. To reduce them, you need to edit the installation system in the main tables of the file and specify the presence of a sound-proofing substrate with the installation area.

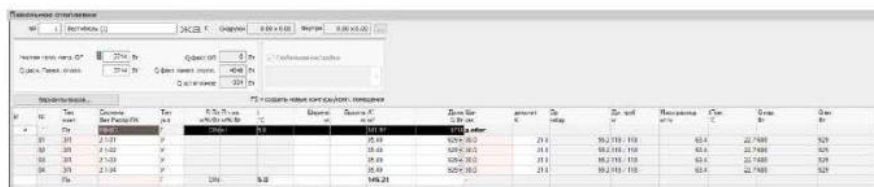


Figure 114-contours suggested by the program for floor placement

The installation step and temperature value can be edited. To draw heating circuits on a drawing, go to Revit, activate **the panel heating/cooling discipline** (Figure 115), select the floor and go to the **structures tab**. First, a zoning plan is created, and then each zone is filled with a floor outline. On the three-dimensional view, you can see which elements were placed.

Thus, the installation of devices ends. Tracing and calculation are discussed in the next section.

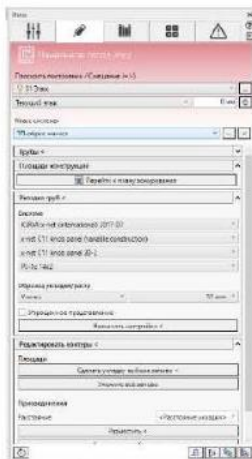


Figure 115-panel settings for panel heating / cooling

If the surface plan does not already exist at the selected level, you must first create a new view that is dedicated **to the panel heating/cooling discipline**. Several commands will be added to the construction area area, which can be used to easily determine the installation areas. Click on **the borders button** to define the area's border lines. It will define the laying area within the specified boundaries, as an alternative. **Area in a room** you can choose one of the spaces and take into account its entire area. The area taken into account in this way can be divided into parts and create several heating circuits in the room.

Using the button **area of premises** and select the required spaces (Fig. 116) by pressing clavishi Esc the command will be finished, and the area is created. Divide the area according to a certain Linear number of heating circuits. The command requires you to select an area and specify the starting side of the division.



Figure 116-indication of zones for placing underfloor heating

Buffet counters are located in the zone. If the squares are prepared accordingly, then Linear Buildings can be added Building. We use the same tool as when placing heating radiators. Figure 117 shows the results of placing floor heating contours on the 1st floor.

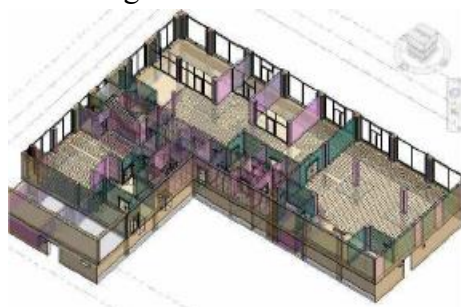


Figure 117-contours of underfloor heating in the 1st floor premises

3.1.4 tracing and calculation of the heating system

This section will describe the method of tracing pipelines, then radiators and convectors will be connected, a heat source will be placed in an individual heat point (etc.) and a distributor for heating circuits will be installed. The floor calculation will be integrated into the calculation via a mixing circuit and an expansion tank will be installed in the network. The tracks and heat source will be connected to the distributor and the network will be equipped with its own shut-off valves.

Tracing. The control panel in **the heating discipline** відкривається opens the first floor-the basement of the house. An ITP is installed at the entrance to the building, a heat source will be installed here, and a heating distributor will be installed on the next wall. The distributor of incomplete heating is located FROM the bathroom on the 1st floor.

On **the structures tab**, you need to specify the design height for laying pipes, for the project it will be 30 mm. Along with various manufacturers' pipeline systems, a number of neutral systems are offered. System of electric-welded steel pipes with diameters of DN25 is selected from the neutral systems (Figure 118).

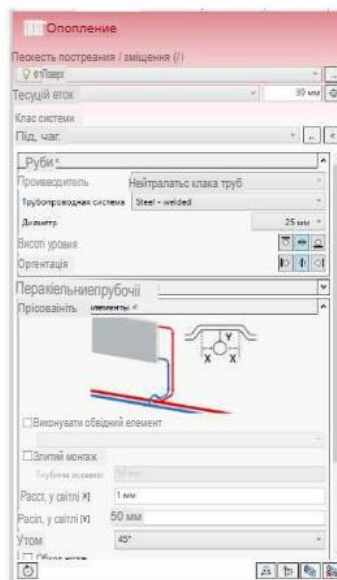


Figure 118 - Selection of pipes for radiator heating

Параметр **The level height parameter** defines the base edge of the pipes relative to each other (Figure 119).



Figure 119 - table of options for aligning the base edge of pipes in relation to one to the second

After clicking on **the pipe button**, a return line is built starting from the radiators of the 1st floor. The family of pipe systems and fittings was loaded before drawing and is available in the drawing.

If there is a need to keep a certain distance from the wall for laying pipelines, then you can set this size. Temporary dimensions will be placed between the pipe axis and the wall.

After that, the pipeline will be moved automatically. If the numbers after the comma interfere with setting dimensions, you can adjust them by clicking on **the units button on the Project tab**.

After building the return line, you need to create a feed line. To do this, use **the parallel pipeline command** in the lower area of the structures tab (Figure 120). A prerequisite for building a parallel pipeline is a pipeline that has already been created. Based on the existing return line, you can use this command to create a route containing various pipeline systems with certain materials and distances. At the same time, these systems can belong to different disciplines.

Welding steel, with DN25 and a distance of 100 mm, is selected as the material for the supply line. You can save and load the pipeline list that you have compiled once at any time.

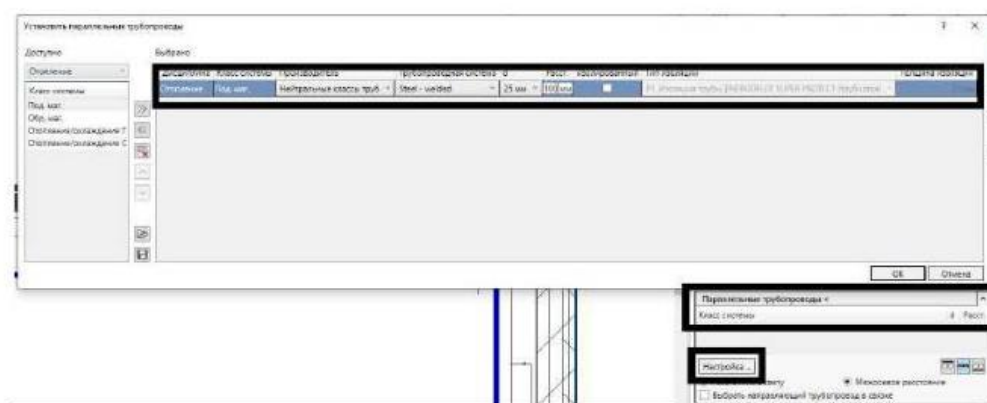


Figure 120-Setting up the creation of parallel pipelines

To create a parallel supply line, select the return line, and then run this command. You will be offered a choice of placement options for the supply pipe: front, rear, and above or below it. The supply line will be drawn behind the reverse one.

Connection of heating system elements. After removing the temporary dimensions, the heating devices can be connected directly to the track. To do this, use **the attach elements command**, which automatically connects elements to the selected pipelines: feed and return.

This command has several options: connection of the wall radiator and its insertion depth, location of pipelines, more visible, above or below a closer one. You can also set the minimum distance between pipelines. If the pipelines are laid in the floor, then choose **ауть** the option of remote location of the pipeline above the Near one.

When connecting, separate fittings or a bypass element can be used. The default liNear Cross Over family contains the corresponding outline elements. In addition, the function offers loaded project families, which could also contain outline elements.

When connecting with concealed mounting, pipes are installed in the wall (fig. 121). In order for the program to determine the location of the walls, the mayor's spaces are used.

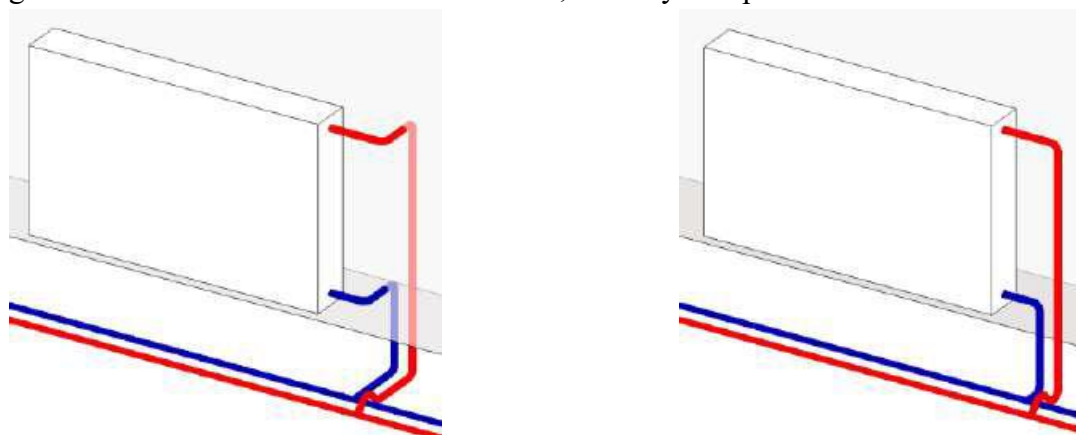


Figure 121-Mounting options

In the case of the activated position, the pipes will be drawn horizontally into the wall from the element, while the depth of insertion into the wall can be changed.



Figure 122 - Setting the insertion depth

Light distance [X] defines the light distance between the outer edge of the main pipeline and the junction bend (Figure 123).

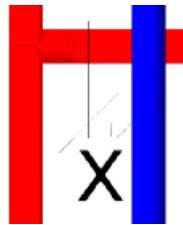


Figure 123 - distance between pipes [x]

Light distance [Y] defines the light distance [Y] between the connecting pipeline and the main pipeline (Figure 124).

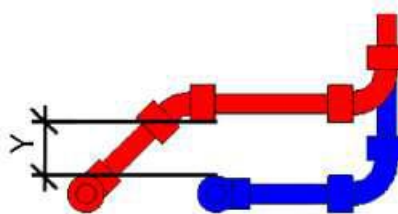


Figure 124-distance in light [Y]

Tap angle. ираютъSelect the tap angle value to use when traversing sections.

Workaround below. If the main pipelines run horizontally, you can determine whether the bypass should run over or under the intersecting pipeline, and also set the distance in light. If the main pipelines run vertically, then a bypass can be performed in front of or behind the intersecting gadfly pipeline (Figure 125).

If the command is activated, the connection will be made under the intersecting pipeline, if the main pipeline is located parallel to the element, otherwise - above the intersecting pipeline.

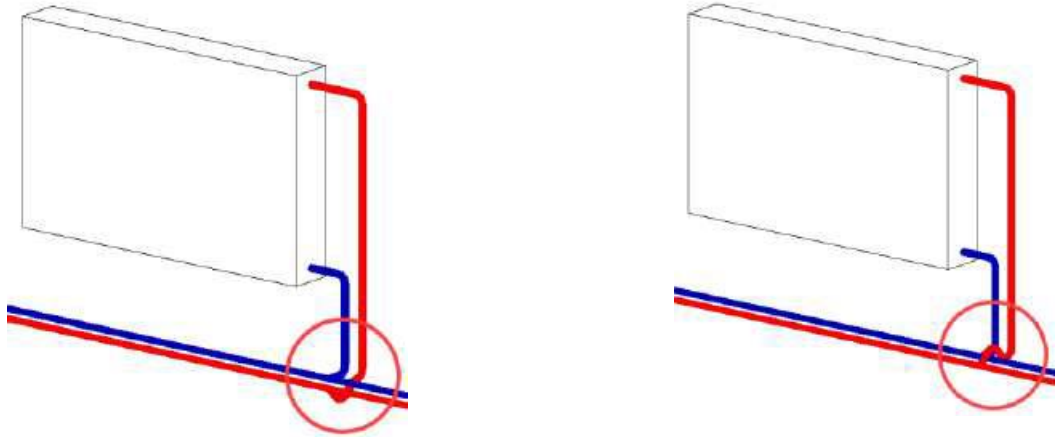


Figure 125-Pipeline layout options

Save the fittings during the crawl. Depending on the spatial situation, when traversing an intersecting main pipeline from above or from below, additional shaped parts may occur. If this option is enabled, the bypass element will be used (Figure 126).

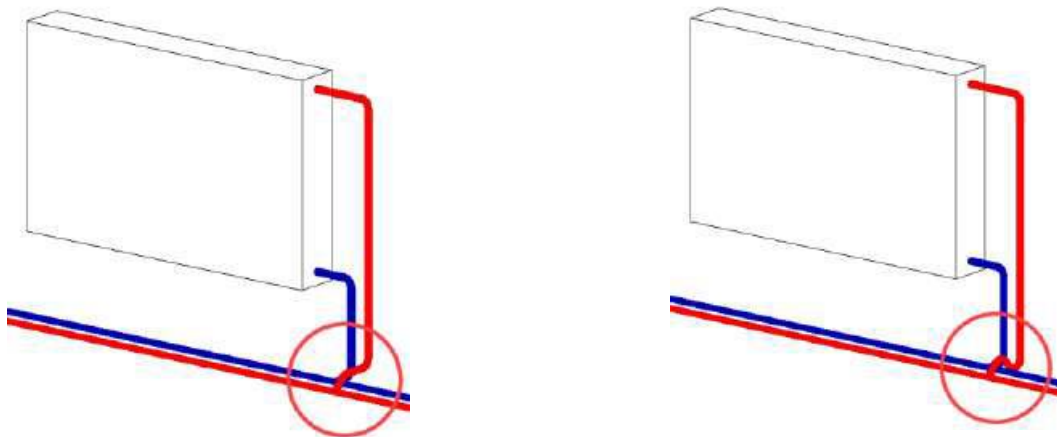


Figure 126-Fitting placement options during the bypass

If the command is activated, the pipes will run vertically down from the element. The pipe that crosses the main pipeline will not be drawn up to its height, but will stop slightly higher.

If the command is deactivated, the pipe that crosses the main pipeline will first be drawn vertically down to the height of the main pipeline (Figure 127). Additional fittings will be required to implement the bypass.

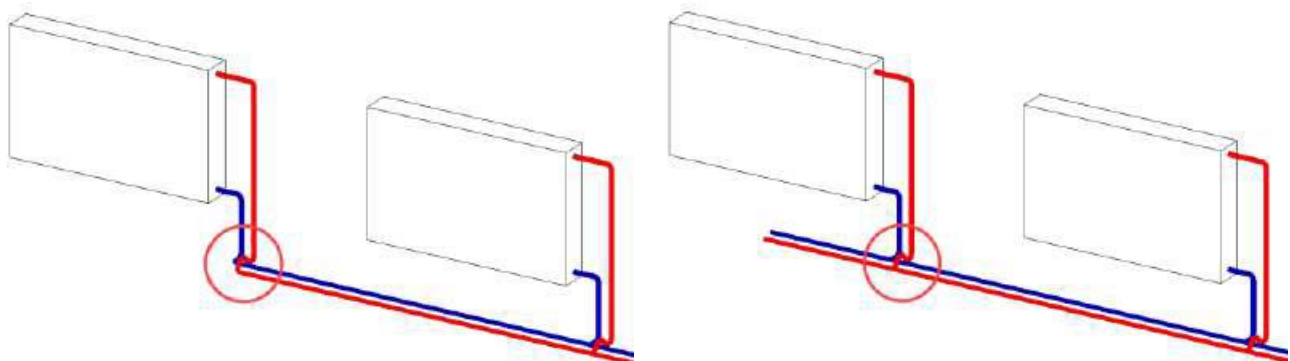


Figure 127-Pipeline connection options

In the deactivated case, the last elements will be attached using tees, and the end of the pipeline will remain open. This makes sense if further construction of the pipeline system is planned.

All radiators on the ground floor will be connected. Next, you can start connecting **внутрішньопідлогових indoor convectors** on the 2nd floor. The connection algorithm is as follows: mark the contour of the return line, specify the distance from the wall to the pipelines, use the parallel pipelines tool, and select the location. You can also use trunk encoding by selecting everything in the clipboard using a filter where the check boxes for pipelines and fittings are selected. Use the **Copy-Paste commands with align to the selected level** to copy pipelines to different levels, for example, on the 3rd floor. If the connection position is changed, for example, from left to right to ensure a perfect connection to the track. To do this, in the LiNear Building program, using the **pipette function**, the PP column changes its position, and the data is transmitted to Revit, and the ECT object will be automatically attached.

Replacement of the heat generator /ETC in the basement.

At the next stage, the placement of the heat generator in **технічному the technical room** on the 1st floor is considered. To do this, on the **heating disciplines tab** in the library, set the design height to 1000 mm. In the section there are three options of heat generators available for loading, choose the floor type (Figure 128). After selecting it, the family will be loaded, and after clicking again, it will be attached to the cursor. By pressing the Spacebar, the element can be rotated 90 degrees.

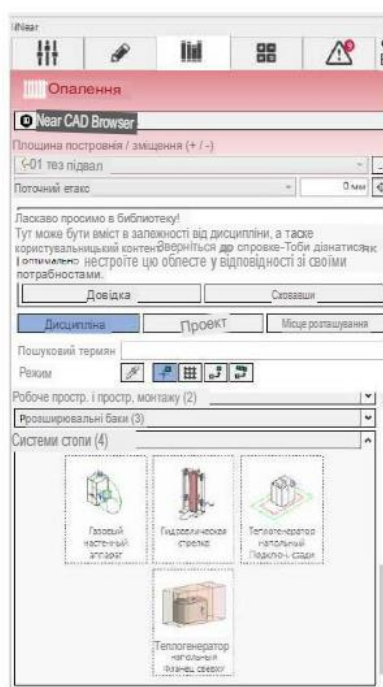


Figure 128-selection of the heat generator

We place the device **виконуємо** according to the technical specification. In the properties of this device, there are several connection options, including size and connection sizes (Figure 129). You can specify the radius of the supply and return pipelines in **the general area**.

To connect the heating system, it is more convenient to work not with a plan, but with a vertical working section. In this view, only some of the attachments are visible, and the rest are

hidden. After a mouse click on an attachment, you can also select the position under/above it, as well as cancel the position. The attachment system is also visible here.

The next step requires allocators. In **the discipline group**, there are four configurators: for neutral heating devices, an apartment-by-apartment dilution station, and distribution manifold configurator. When we select the collector, we see that none of the collections are loaded yet. First, you need to set the distributor pipe types.

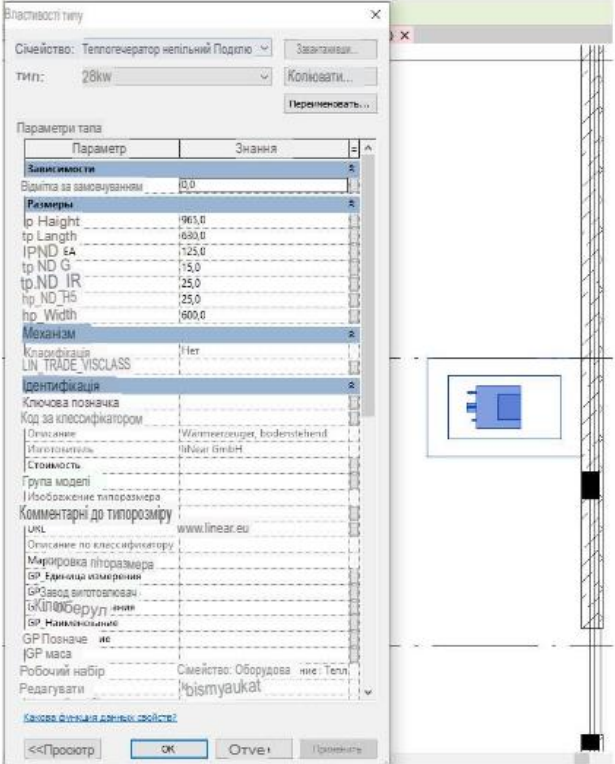


Figure 129-Generator type properties

Depending on the pipe types, the appropriate diameters are available. You must also specify the type of distance: light distance or center-to-center distance; add four pairs of fittings with a diameter of 32 for floor heating devices, and one pair of fittings with a diameter of 40 for the heat generator. The distance in the light between the fittings should be 200 mm (Fig. After clicking OK, you must assign a name to the family. After that, the cursor will be placed for insertion into the drawing.

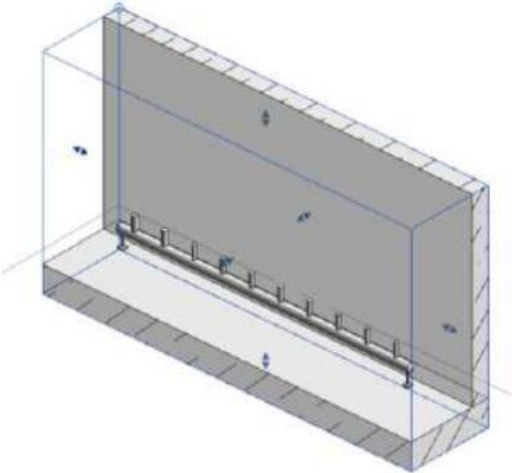


Figure 130 - Setting parameters for heat distribution

To place the underfloor heating distributor, open the LiNaer Buildig programLiNaer Buildig, **view** branches of pipe and wires tab. All branches of the heating system are created in the column, each of which can be designated with a name and Number for theheating and underfloor heating systems (Figure 131). Floor polygons are attached to the collector when you click **Draw distributors**, which will be built in Revit.

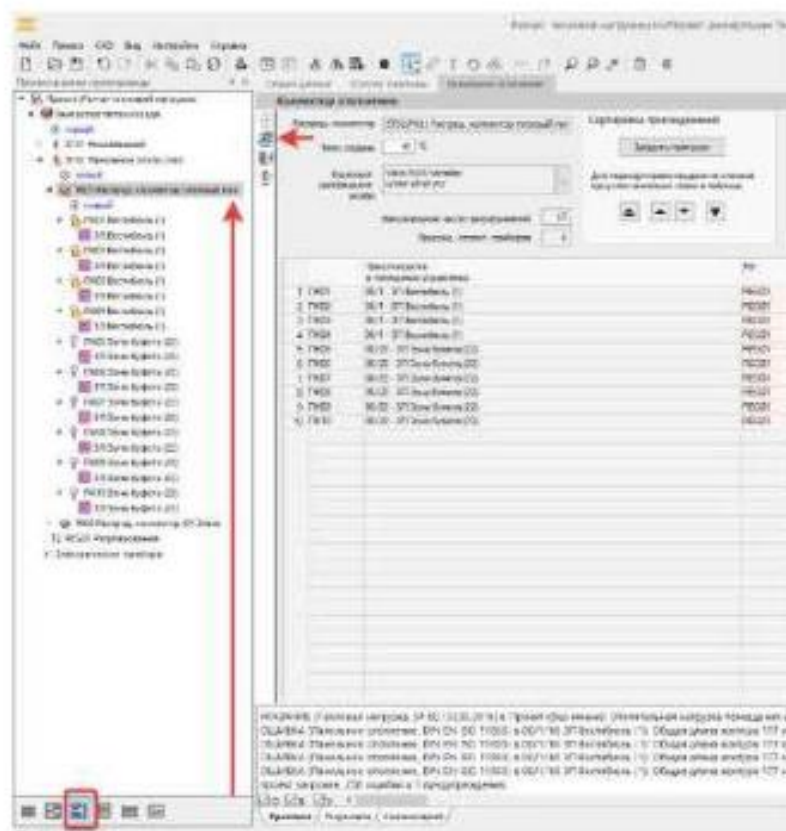


Figure 131-distributor for underfloor heating

In the distributor properties, all the necessary parameters are set for individual heating circuits, and in the data area, the transmitted capacities, mass flow rates, and pressure losses are shown.

Next step: connecting the generator, distributors and heating circuits. It is necessary to draw the pipeline in the direction of the boiler. The finished connection diagram in 3b form is shown in Figure 132.

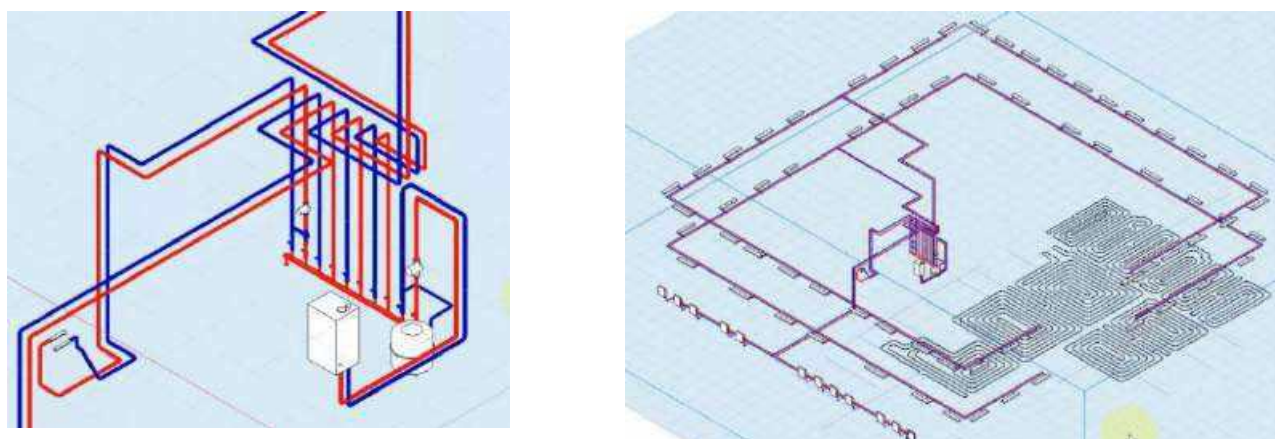


Figure 132 - Diagram of connecting the generator to the distributor

Traversing intersections

If pipelines intersect with each other with some element or architecture, then you can use the traversal tool to solve this situation (Figure 133).

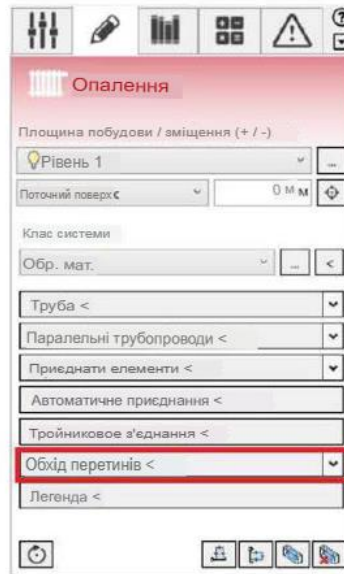


Figure 133-Section bypass point

To configure the parameters for solving this problem, you need to:

- Open **the section traversal area** (Figure 134).

-



Figure 134-Viewing the intersection area

- Specify the horizontal and vertical minimum distances (distance X, distance Y).
- Select the angle of the bypass knee.
- Use the enable desired crawl parameters check boxes.
- Activate **the element traversal mode**.
- Select the pipelines to be bypassed and confirm by pressing Enter.
- Select the items to be crawled and confirm by pressing Enter.

Optimal solutions will be proposed (Figure 135).



Figure 135-Crawl placement options

You can view the options using the View button when you determine the appropriate method, the selection is confirmed by clicking Accept. As a result, the selected method will be applied and the intersection situation will be resolved (Figure 136).

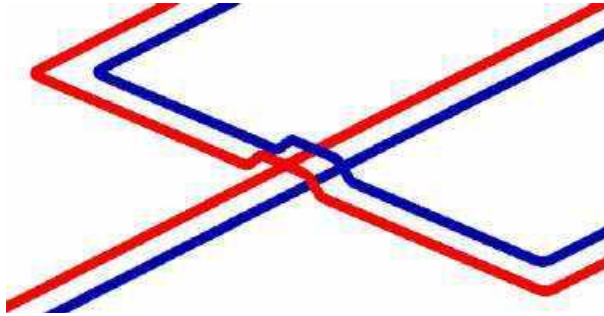


Figure 136-solution to the cross-section problem

If the option is activated, existing fittings will be rotated if necessary and used to bypass cross-sections. This avoids the use of additional fittings. The X distance is not used in the example above, because the fittings on both sides are used for traversal. If the option is not activated, additional fittings will be added when traversing sections.

The insulation accounting function allows for x and y light distances in the case of insulated pipes and ducts to be specified both in terms of insulation and in terms of pipelines or ducts (Figure 137).

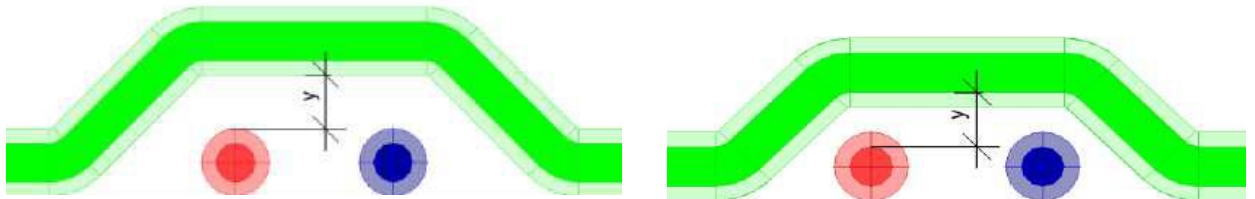


Figure 137-Activated and unactivated commands

When the bypass intersections with insulated pipelines command is activated, the X and Y distances are specified relative to the insulation of pipelines and air ducts. When the bypass cross-sections with insulated pipelines option was not activated, the X and Y distances specified relative to the pipelines and air ducts themselves were not taken into account.

You can adjust the bypass manually, for example, when pipelines intersect with an element (isometry, top view) (Figure 138).

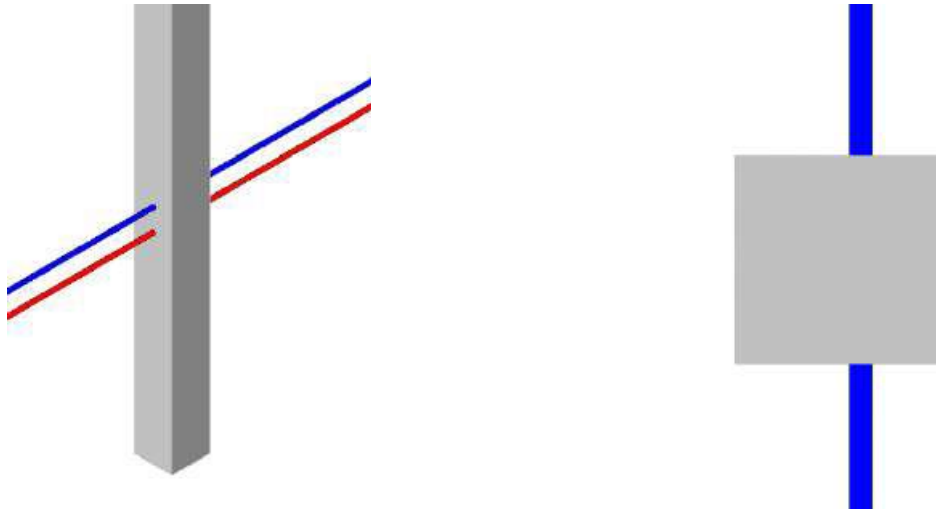


Figure 138-section of pipes with a column

You need to open the cross-section bypass area, select the angle of the bypass elbow, activate the manual bypass mode, click on cross-section bypass, select pipelines that intersect the object, and confirm by pressing Enter.

A message will appear explaining the procedure (Figure 139). Clicking the mouse will indicate the length of the crawl. The base points indicate the length of this segment to be traversed. You can specify any length, and the minimum length will be met automatically due to design restrictions. Use the mouse to indicate the distance that should be observed between the element and the crawl, and *лівою кнопкою* confirm the selection with the left button. The third mouse click shows the X-axis distance in the top view. In the section view, the third mouse click shows the distance along the Y-axis. Due to design constraints, the minimum length will be met automatically. The intersection situation has been resolved.

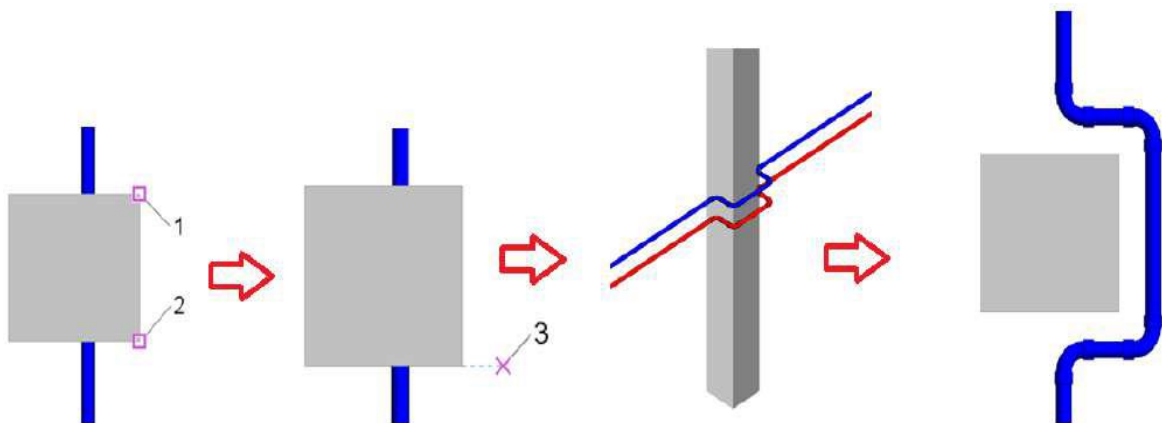


Figure 139-steps to correct the situation of bypassing the column

The next option is to bypass element intersections in linked files. If pipelines or air ducts intersect with link elements, you can use the bypass tool to resolve this situation. Repeat the first steps of the previous method to select the activation **of the crawl mode for elements** in a bundle. *Вибір* They will select the pipelines for which the bypass should be implemented and confirm it by pressing Enter. *Вибір* Select the elements in the bundle that you want to avoid intersecting with, and *жувуть* confirm with the Enter key. Various options will be shown.

As a result, sections with pipelines were bypassed in conjunction with VC networks (Fig. 140).

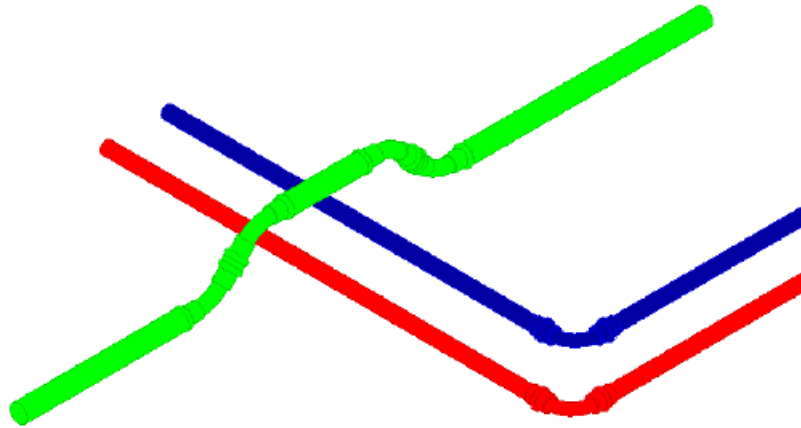


Figure 140-Result of constructing the VK pipe bypass

As a result of work on equipping the model with a heating system, it was possible to fully calculate the building. It was possible to calculate the building's thermal parameters with an accuracy of up to 90% compared to manual calculation in MS Excel. As a result of this calculation, a detailed 130-page report was provided on the values of enclosing structures that affect the building's heat loss parameters. This report can be useful not only for the OS designer and компанії heat supply company about the load, but also for the customer to assess the energy efficiency of the building.

As a result of the calculation itself, errors are detected in ARS or families that accidentally or inattentively ended up in the project, correcting which improves the information model, makes it clean and up-to-date.

The duration of calculating the load on a building in the LiNear Building program takes up to 30 minutes of working time, provided that the program is mastered and the AR model is correct, which in turn is up to 7200% (4 working days) faster than with manual calculation. Distinguishing the function of the program to select different heating options, from radiator to underfloor heating, allows you to view the object from several angles at once for both customers and designers. All physical parameters, such as pressure loss, velocity, temperature in pipelines, etc., are taken into account and displayed simultaneously.

Cross-section traversal functions allow you to work with cross-sections in several clicks, because when an error is detected on a cross-section with something, ready-made solutions to the problem are provided.

The program is fully adapted to Ukrainian legislation and is based on the current rules and regulations. Its functionality is difficult and not intuitive to master. You will need to spend a lot of time to fully switch to using it. This section covers all the technological steps that can help you learn and work in the program. The authors recommend a program for designing heating systems using information modeling technologies.

4.2 Equipping the model with a ventilation system

The process of equipping a building model with a ventilation system consists of several stages:

- Characteristics of premises
- Placement of grilles and air ducts.
- Connecting elements of the ventilation system to each other.
- Calculation of the ventilation system.

- The process of calculating the ventilation system.

Creating types of plans for the ventilation system, creating spaces, and zoning were discussed in the first part of preparing the model. You can immediately proceed to the characteristics of the premises, design and calculation of the ventilation system.

An additional characteristic of the ventilation system calculation rooms is the number of people. This parameter is filled in for all spaces for allocating the required spaces, and then in the properties of the energy consumption calculation section. Hold down the Ctrl, key to select office spaces and set the number of people in them that is pre-defined in the terms of reference. So, for example, room number 8 - auditorium - 150 people, Office-30 people, dressing room - 2 people, and so on.

3.2.1 Duct design and connection to the system

Go to **the ventilation discipline on the Library tab** of the control panel. The room design will use a combination of air vents that will allow air to pass through the upper vent and exhaust air through the lower vent. These special elements are already contained in the library. On the address panel, you can select the height of the grid and the combined grid that will be installed in all rooms in the middle above the door. Exhaust valves of sanitary facilities will be placed on the ceiling (fig. 141).

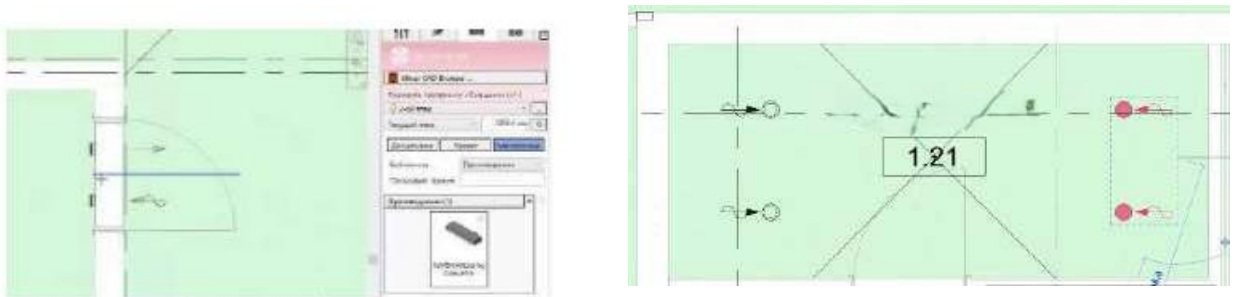


Figure 141- Options for placing grilles in the office and bathroom

In the **Plot / Offset plane (+ / -)** area, select the base and reference planes. If the base and reference planes match, the current floor is selected. The offset to the reference plane is specified. If the element must be positioned below the reference plane, a negative offset is entered. Specify **the system class** and open **the channels area** (Figure 142).

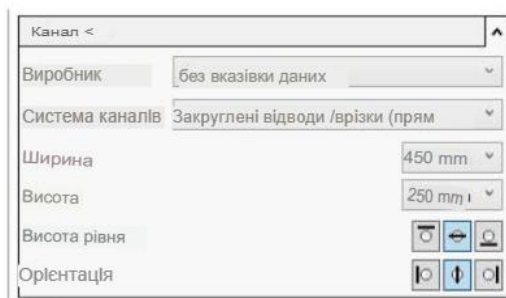


Figure 142-Control panel for designing air ducts

You must choose: manufacturer and channel system; diameter (for round channels) or width and height (for rectangular channels); channel level height: top edge, center or bottom edge; duct alignment option: left edge, middle edge or right edge.

If you need to create vertical pipelines during design, then enter the desired height in the options bar in the offset field (Figure 143). A vertical pipeline will be created and the design of a mountain of zonal pipelines can be continued (Figure 144).

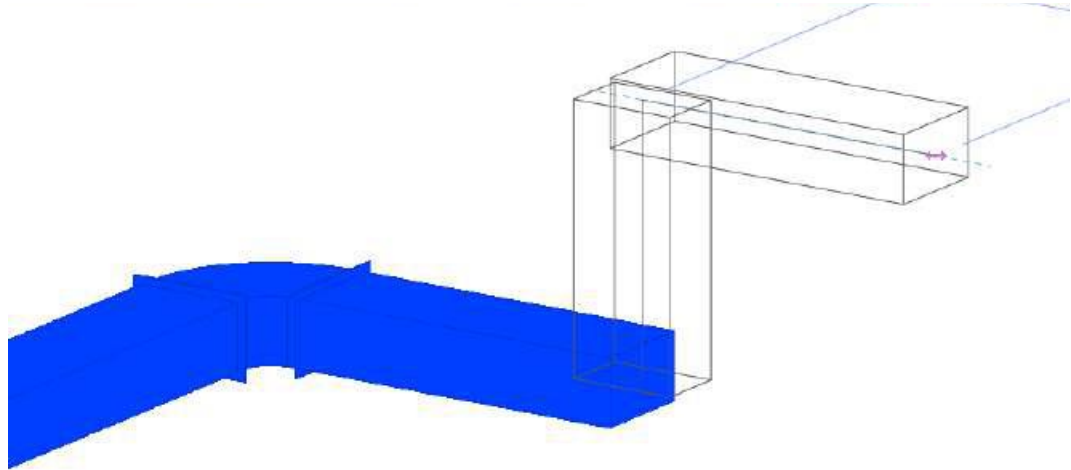


Figure 143-Setting the offset

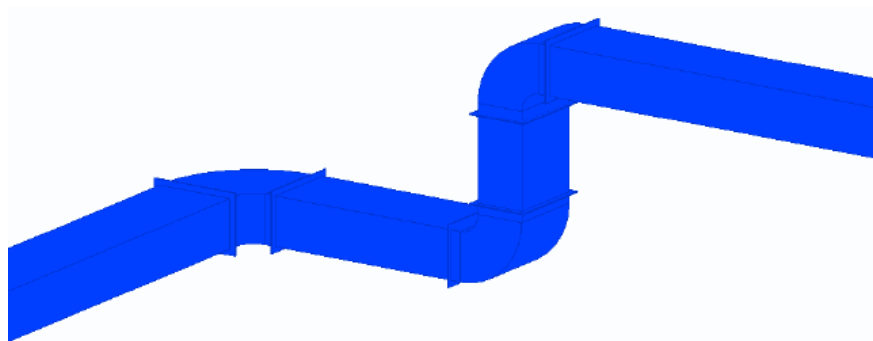


Figure 144-Result of the designed air ducts with the selected system and material

Attaching elements

To do this, use the command to automatically connect elements to the selected air ducts. Connection to the main air duct can be made as a tee or using a fitting (Fig. 145).

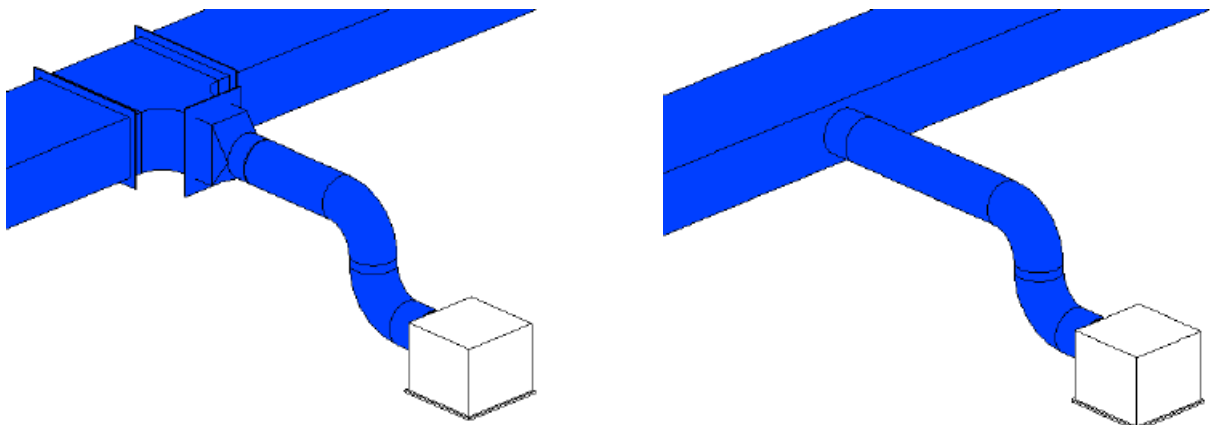


Figure 145-connection options using a tee or fitting

Flexible connection of the flow port is also possible (Fig. 146).

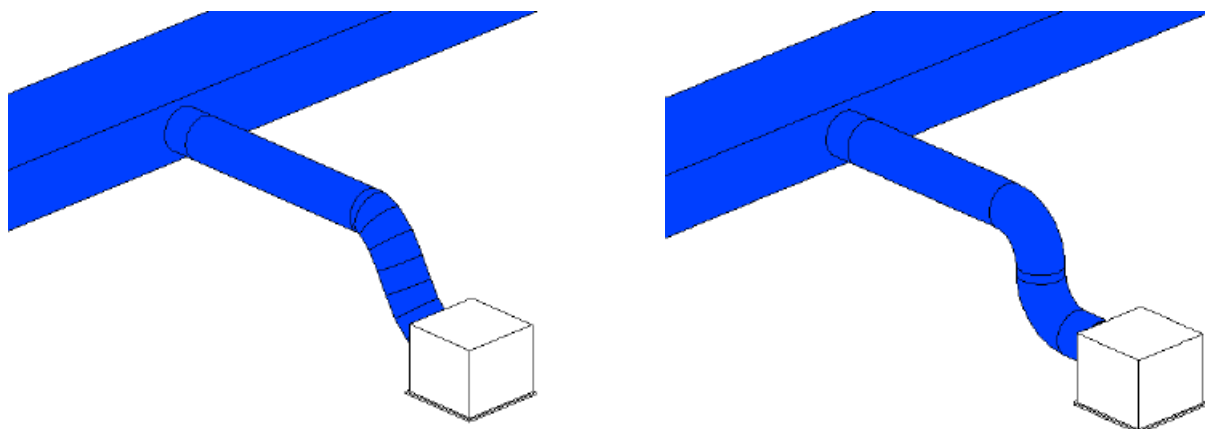


Figure 146 - Activated and deactivated flex command

If the command is activated, the first part of the connection from the exhaust port will be a flexible air duct. Additionally, the length [L] is defined. If you enter a length value of 0 mm, this part of the duct will be fixed. If you specify a length that exceeds the distance to the channel, all connections will be made using a flexible duct. In this case, no plug-in parts will be added.

If the *деактивованої* option is deactivated, the connection is made entirely of fixed air ducts.

Tap on the last elements

When activated, the last elements of the ventilation system will be connected via taps, so the ventilation system will be finished (Fig. 147).

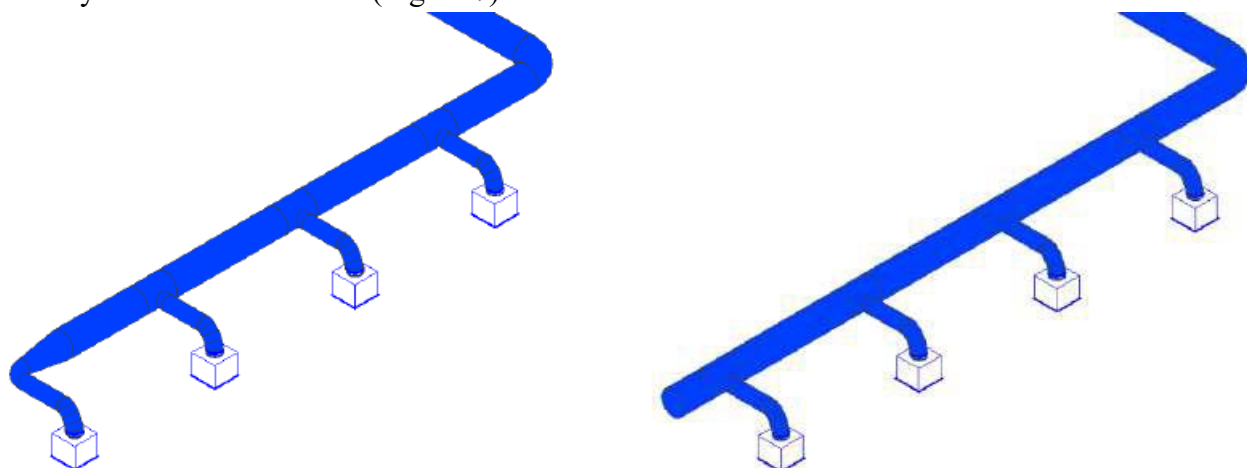


Figure 147-The result when activated and deactivated to the tap creation command

In the deactivated state, the last elements will be connected using tees, and the end of the duct will be open. This makes sense if further construction of the ventilation system is planned.

After designing the supply and exhaust air ducts, even without a supply air unit, it is already possible to perform calculations using additional elements. To do this, you need to add hardware for the beginning and end of the network. you don't need a fan for the calculation.

For outdoor air, the outlet **вихід**startof the network is selected, and for supply air, the endpoint is selected. Since the beginning of part of the network corresponds to the fan, which is the building's engineering equipment, it does not take over the system when installed in the pipeline. inityazhny for the supply air (fig. 148). To avoid errors in further calculations, you must also specify the volume costs.

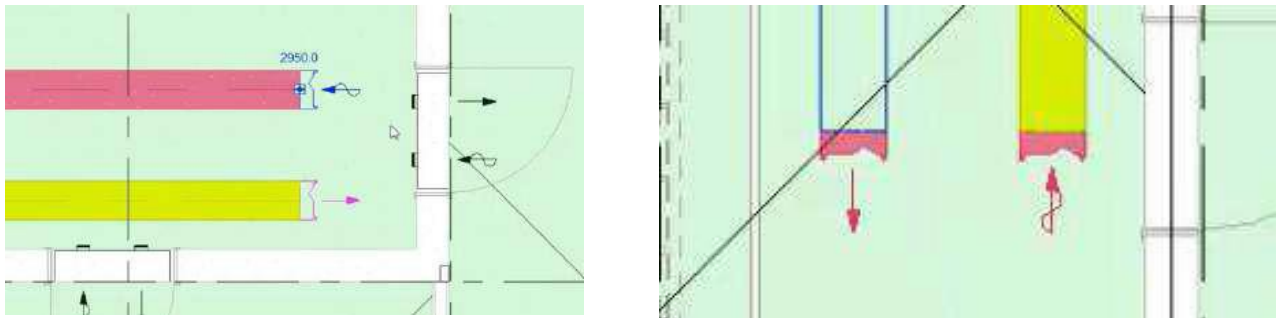


Figure 148-Elements at the beginning and end of the system

3.2.2 calculation of the ventilation system

Now you can check whether the original dimensions of the air ducts are correctly selected, this can be done in **the analysis - configuration tab**. Before you start calculating the ventilation network, you must perform an analysis of all sections on the model. It will be started automatically by clicking **the calculate button**. Only those elements whose system classes are specified in **the Configuration dialog will be analyzed** (Figure 149).

To analyze the geometry of the ventilation system, first of all, individual sections will be calculated, bounded by tees or elements in which the pipe size changes, for example, in adapters.

After that, the program will search for the initial element of the ventilation network. This element is the fan or the beginning of a part of the network. If there is no fan, the program searches for the beginning of the network part.

The program creates an installation connected to a single source element. The fan creates two settings, and the beginning of the network part creates one. The fan and the start part of the network should not be installed in the same flow path. The installation type is determined according to the system of the First channel after the initial element.

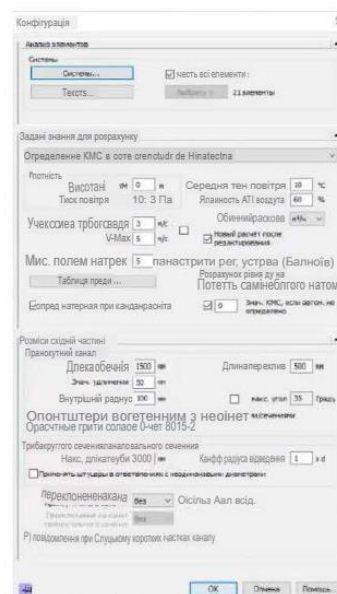


Figure 149-configuring the configuration for calculation

Flow paths in the ventilation system are determined based on the initial element to one external protective grid. However, exhaust and supply air installations only need one fan per installation.

The results of calculating the noise level are shown in the order of its occurrence: external protective grilles noise to the outside, the result of the acoustic calculation for the flow path with an external protective grating refers to the environment. Passageways make noise in the rooms where they are installed, so their results are displayed relative to the corresponding room.

Before calculating the ventilation network, you can set some parameters in **the Configuration dialog box**, such as speed limits and friction pressure loss.

At the beginning of the calculation, the smallest diameter will be selected for all sections of the ventilation system with unknown diameter values. The smallest diameter is the first one specified by the diameter check box in **the channels and pipes dialog**.

First of all, the program determines the required volume flow rates in the ventilation network based on the volume flow rates of the intake openings and the ends of the network part from the supply and exhaust air installations. The total volume costs are transferred to installations of removed atmospheric or air. If several external protective grilles are connected to the fan, the volume costs will be divided as a percentage on each corresponding branch located behind them. This primary assignment of air volume flow rates can be changed by the user. To do this, uncheck the **calculated field** in the data dialog for the external grid element or the beginning of the network part.

If the inlet or end of a part of the supply or exhaust air system network contains only power data, the volume flow rate is calculated based on the set supply and exhaust air temperature.

The flow resistance in the ventilation network is calculated from the volume flow rate of air. If the limit values for friction resistance or speed are exceeded, the program assigns such diameters to the corresponding sections, if they were fixed. At the same time, the program is guided by data from the settings **of the channels and pipes window**.

The most unfavorable flow path in the ventilation network is automatically recognized by the program, since the program first detects pressure losses in all flow paths. This unfavorable flow path will be marked in color in **the flow path table in the Calculation dialog** (Figure 150). When you press the U key on the keyboard, the flow path is selected in the table.

The tables show all flow paths, sections, and elements of the calculated ventilation system. The table shows the calculated data of all flow paths available in the selected installation. This allows you to view all the finite elements available in the network. When you select a flow path, all relevant sections are shown in the section table.

The most unfavorable flow path will be marked with a color. You can select the most unfavorable flow path by clicking on U.

№	Тип	Пропускний отвір	Об'ємний р	Др-Трубоп	Др-Місце	Др-Баланс	Др-Загальн
			м³/год	Па	Па	Па	Па
14	Вит	Тарілчастий клапан		15	21	55	132
15	Вит	Тарілчастий клапан		15	21	55	132
16	Вит	Тарілчастий клапан		15	22	57	132
17	Витягнутий	клапан		15	22	58	132
18	Витягнутий	клапан		15	23	60	132
19	Витягнутий	клапан		15	23	60	132

№	Тип	T	V	l	a	b	d (dh)	v	W	I''R	ζ	Z	ΔP	ΔP _W	ΔP _A
		°C	м³/год	м	мм	мм	мм	м/з	Па/м	Па		Па	Па	Па	Па
1	Вит	20	861	2.25			280	3.9	0.62	1.4	0.30	2.7	4.1		4.1
2	Вит	20	861	2.58			280	3.9	0.62	1.6	0.30	2.7	4.3		4.3
3	Вит	20	681	1.40			250	3.9	0.71	1.0	1.86	16.6	17.6		17.6
4	Вит	20	645	1.06			250	3.6	0.64	0.7	0.11	0.9	1.6		1.6
5	Вит	20	627	2.14			250	3.5	0.61	1.3	0.06	0.5	1.8		1.8
6	Вит	20	558	1.72			250	3.2	0.50	0.9	0.37	2.2	3.1		3.1
7	Вит	20	486	1.61			250	2.8	0.39	0.6	0.26	1.2	1.8		1.8
8	Вит	20	486	0.58			200	4.3	1.13	0.7			0.7		0.7

№	Тип	Ім'я	l	a	b	c	d	R	I''R	ζ	Z	ΔP	ΔP _W	ΔP _A
			м	мм	мм	мм	мм	Па/м	Па		Па	Па	Па	Па
8	R	Wickelfalzrohr (Кругла...	0.66				280	0.62	0.4			0.4	0.4	0.4
9	OTP	Rohrbogen					280			0.15	1.4	1.4	1.4	1.4
10	R	Wickelfalzrohr (Кругла...	1.08				280	0.62	0.7			0.7	0.7	0.7
11	OTP	Rohrbogen					280			0.15	1.4	1.4	1.4	1.4
12	R	Wickelfalzrohr (Кругла...	0.84				280	0.62	0.5			0.5	0.5	0.5

Figure 150-Table of ventilation system characteristics

The table shows the calculated data of all sections available in the selected flow path. When you select a plot, all elements related to it in the elements table will be shown. The table shows the calculated data of all the elements available on the selected site. Functions for calculation and tables related to the entire ventilation system are presented in Table 1.

Table 1-Calculation functions

Element	Meaning
Installation Values element	If a project contains multiple installations or parts of a network, you can choose which installation or part of the network should be shown in the tables in the calculation dialog. In the Installations list, you can select only external systems or only indoor systems.
-	This button opens the technical data of the initial element. Here you can also change the name of the installation that appears in the drop-down list.
Calculate	if you have made any changes, run the pipeline network calculation to see the current results.
Visualization	This function shows the properties and results of the pipeline network calculation in color on the model
the result	shows additional information about the pipeline network, as well as the calculation results that are printed out.
the report	shows an overview of warnings and errors that occurred during the calculation. You can find out whether this is a warning or an error by using the button color:

	<ul style="list-style-type: none"> - white: no errors or warnings. - orange: the report contains warnings. - red: there are errors in the calculation.
Label	Entries with the results of calculating network elements and sections.
Update	if elements or sections are already signed, but the calculation results have changed, you can update the label and fill it with up-to-date data.
Adjust nominal diameters	if the calculated nominal diameters differ from the designed ones, you can transfer the calculated diameters to the model. Adjustments to nominal diameters can be made for the entire network as well as for individual parts of it.
Pressure loss, Noise level	Switch button displays the results of pressure loss calculation or acoustic calculation. The flow path table shows the pressure loss or acoustic calculation results, respectively.

Now you can create a ventilation unit by selecting installation A in the ventilation configurator and creating a new component.

You can set any configuration of the supply air unit, for example, a chamber with a fan, the chambers themselves, etc., from the selection list from suppliers. The upper and lower levels are also indicated there, as well as the width and height abarites (Figure 151). After clicking OK, you can enter the name of the installation. Then, by removing the above temporary installation simulation elements, you can connect the air ducts.

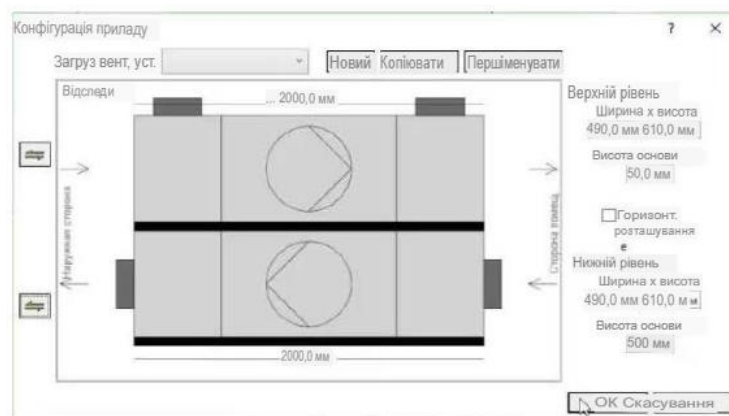


Figure 151-configuration of the supply and exhaust system

Output of results. From the calculation program, you can get data in the form of a selection report, which reflects the list of materials of the calculated installations, volume costs, fan pressures, noise level, and a balanced list of pressure losses of all flow openings (Fig. 152).

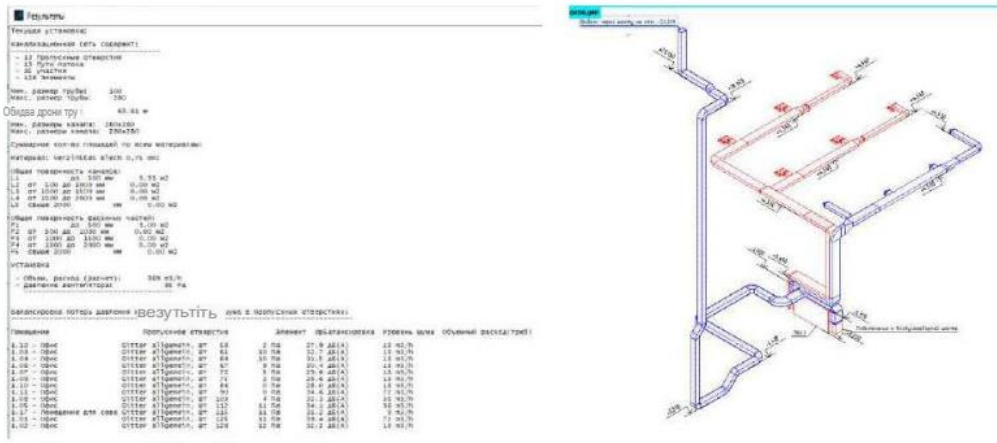


Figure 152-final calculation report and designed ventilation system

As a result, Linear is capable of many useful functions for automating the design of ventilation systems. The aerodynamics of the ventilation system was calculated, and the supply and exhaust system equipment, channels, and grilles were selected.

Linear cheats a built system, understands its physical parameters, sees broken data and flow paths, suggests recommended dimensions for systems, and visualizes the data obtained. defines what can be attached to each other in different ways.

Also, as in the heating section, a useful tool in the program is the ability to analyze for collisions, as well as correct them. All systems, elements, and connections that pass nearby are read, and you are prompted to bypass them in a couple of clicks. This section covers all the technological steps that can help you learn and work in the program.

Conclusions about the features of the LiNear app

This software is a full-fledged replacement for all programs that software designers use in their work. LiNear is able to adapt to A's standards and regulations, making this software unique in the market today. The program library has a huge database of equipment from different manufacturers, and simple families with low LOD, which simplifies work and reduces the load on the computer. A big disadvantage is that LiNear is quite complex and intuitive to learn. To fully switch to using it, you need to spend a lot of time.

The main and main advantage is the ability to simulate all physical processes, which saves time for creating a project, performs the most accurate calculation. The obtained data is available to all project participants, which allows related sections to also increase work efficiency, and the customer and the company manager to receive ready-made reports on calculations.

5. COMPARATIVE CALCULATION OF REINFORCED CONCRETE BEAMS ACCORDING TO UKRAINIAN (STATE BUILDING STANDARDS) AND EUROPEAN (EN 1992-1-1) STANDARDS

Shekhovtsov V. I., Malakhov V. V.

INTRODUCTION

Ukraine's integration into the European technical and economic space poses new challenges for the construction industry, the key one being the harmonization of the regulatory framework. The transition from national State Building Standards (DBN) to the European standards system (Eurocodes) is not just a technical update, but a strategic necessity. This process promotes international cooperation, ensures the compatibility of design solutions on the European market, and raises overall standards of safety and reliability of structures. The implementation of Eurocode 2, which regulates the design of reinforced concrete structures, in the form of the national standard State Standards of Ukraine-N B EN 1992-1-1:2010, is a central element of this transformation.

The fundamental difference between the two systems is their philosophy of ensuring the reliability of structures. The approach laid down in State Building Standards V.2.6-98 is based on the use of predetermined “design resistances” of materials, which already take into account certain reliability factors. This method, which comes from a deterministic tradition, offers a direct path to calculation, but the transparency of the safety factors embedded in it is limited.

In contrast, Eurocode 2 uses a semi-probabilistic method. It operates with the concept of ‘characteristic strength’ of materials (f_{ck} for concrete, f_{yk} for reinforcement). To obtain the calculated strength values, these characteristic values are divided by explicit partial reliability coefficients for the material (γ_c for concrete, γ_s for reinforcement). This approach makes the concept of safety more flexible and adaptable.

The aim of this task is to perform a parallel, step-by-step calculation of a typical reinforced concrete beam under bending according to both regulatory systems. This practical exercise will allow you to understand the theoretical differences and quantitatively assess their impact on the final design result, namely the required area of working reinforcement (A_s).

1. THEORETICAL BASIS AND MAIN ASSUMPTIONS

Both State Building Standards and Eurocode are based on a single design philosophy using the limit state method. This method aims to ensure that the structure will not become unusable throughout its design life. The verification is performed for two groups of limit states: the first group (Ultimate Limit State, ULS), which is related to destruction or loss of load-bearing capacity (safety), and the second group (Serviceability Limit State, SLS), which concerns functionality and user comfort (deflections, cracks). This case study focuses on the calculation of the first group of limit states (ULS).

Despite the common goal, the formats for ensuring reliability in both systems differ significantly.

1.1 STATE BUILDING STANDARDS METHOD

This approach uses ‘load reliability factors’ (γ_{fm}) to increase the characteristic (standard) load values to their design values. Furthermore, predefined ‘design supports’ from tables are used for

materials. These values are essentially characteristic strengths that have already been reduced to take into account the (implicit) material reliability factor and other factors, such as the concrete service condition factor γ_{b2} . The basic condition for strength verification is as follows:

the effect of design loads \leq the bearing capacity determined by the design supports.

1.2 EUROCODE 2 METHOD

This approach is more detailed and transparent. To determine the design effect (E_d), partial factors for actions (γ_G for permanent actions, γ_Q for variable actions) are used. The characteristic strengths of materials (f_{ck} , f_{yk}) are determined separately, based on the statistical 5% fractile, i.e. the value below which no more than 5% of test results are expected.

To obtain the design strengths (f_{cd} , f_{yd}), these characteristic values are divided by explicit, clearly defined partial reliability coefficients for the material: γ_c for concrete and γ_s for reinforcement. The formula for the design compressive strength of concrete is central to this approach:

$$f_{cd} = \alpha_{cc} \cdot f_{ck} / \gamma_c.$$

The main condition for verification is as follows:

$$E_d \leq R_d,$$

where R_d – is the design resistance of the cross-section, calculated on the basis of the design strengths f_{cd} and f_{yd} . Separating the coefficients for loads and materials allows for more flexible reliability management. For example, Eurocode 2 sets standard values of $\gamma_c = 1.5$ and $\gamma_s = 1.15$, but also provides a methodology for reducing them, provided that guaranteed and documented increased quality control in production is in place. For example, by reducing deviations in the geometric dimensions of the cross-section and the position of the reinforcement, as well as by reducing the variability of concrete strength, the coefficient γ_c can be reduced to 1.4. This creates a direct economic incentive to improve the quality of construction work, as it allows for the design of more economical structures (with smaller cross-sections or less reinforcement) without reducing the target level of reliability. In the DBN system, where design supports are fixed tabular values, such a flexible, risk-based approach is practically absent.

2. PRACTICAL TASK FOR STUDENTS

2.1 CONTEXT

A Ukrainian construction company is launching a project to rebuild housing destroyed by hostilities in Pokrovsk, Kherson Region. The project is being financed with the participation of a European investment fund. In accordance with Ukrainian legislation, the project documentation must be developed in accordance with current building codes in order to pass state examination and obtain a building permit. At the same time, investors, guided by their own risk management protocols and the requirements of international insurance companies, require parallel verification of key load-bearing structures in accordance with Eurocode standards. This situation creates a real practical need for engineers who are competent to work in both regulatory systems.

The student is asked to act as a design engineer responsible for designing a typical floor beam in this residential building. The task is to calculate the beam (Fig. 3.1) for the first group of ultimate limit states (ULS) under the action of a bending moment.



Fig. 2.1 Restoration of damaged residential building in Pokrovsk, Kherson Region

Task: Perform and document two parallel calculations to determine the required area of longitudinal tensile reinforcement (A_s):

1. Calculation in accordance with the requirements of State Building Standards V.2.6-98 and related standards (in particular, State Building Standards V.1.2-2:2006 for loads).
2. Calculation in accordance with the requirements of State Standards of Ukraine -N B EN 1992-1-1 (Eurocode 2).

The final result of the work should be a comparative report containing a description of methodologies, detailed calculations using LIRA-CAD software, summary results and analytical conclusions. The initial data for this exercise are contained in Appendix.

2.2 POSSIBLE ALGORITHM FOR SOLVING THE PROBLEM


Immediately after launching the SAPFIR PC LIRA-CAD module, the program interface elements become available: command menus, icons, toolbars, service windows with tree lists, graphic windows, etc.


• To set the coordinate axes, use the “Axes” menu command in the “Creation” tab. Use the “Options” icon to activate the “Coordinate Axes” dialogue box.

Select the grid type: “Rectangular”. Set the reference point and its coordinates. The “Intervals” group provides two tables that can be used to set the intervals between the axes and their positions in two directions. To add an axis to the grid, click “Add interval” under the

corresponding table. Click the “OK” button: the dialogue box closes and the programme creates a grid of coordinate axes, which we can see in the graphics window. Use the left mouse button to specify the location of the created grid of coordinate axes.

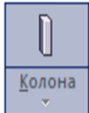
There is a special tool for constructing each element of the structure. It is convenient to call up the tool using the buttons on the toolbar.

 Select the “Wall” tool by clicking the “Wall” button in the “Create tab”, which will display the “Wall” tool properties panel at the top of the programme window. Select the method for constructing the centre line: “Straight Segments”, “Chain”, etc. Use the “Material” drop-down list in the construction properties window and select the wall material. It is important to note that for internal partitions, you should set the value “Partition” in the “Function” drop-down list and the value “Load” in the “Interpretation” drop-down list. Set the thickness depending on the material, the base level, and the method of attachment to the axis. Using the three-dimensional locator controlled by the mouse, enter several points of the axis line in sequence.


 Select the “Door” tool from the “Tools” panel. The door properties are loaded into the tool properties panel. The dimensions of the opening, the depth of the door leaf (the distance from the wall plane to the door installation plane), and the level relative to the floor can be specified in the corresponding editing windows. Openings are placed on walls using a three-dimensional locator. The locator determines the position of the base point of the opening in the model. The opening can be tied to its base point in one of three ways: point on the left, point in the middle, point on the right. The binding method is switched using the icons on the “Door” tool properties panel. The filling of the opening can be selected using the corresponding dialogue box. In the “Fill opening” dialogue box, specify the elements for filling the door opening, select a ready-made door leaf model from the library and confirm your selection with the “Fill opening” button.

Use the locator to place several openings on the building facade. To do this, place the cursor on the wall image, move the “mouse”, control the locator, find the desired position, and fix the opening with a single click of the left “mouse” button.

The algorithm for adding windows to a building model is identical to the algorithm for adding doors described above. Select the “Window” tool, configure the basic window parameters, specify the opening fill, and use the 3D locator to place the door opening within the wall of the building model.

 Select the “Column” tool. Click the ‘Cross-section’ button and set the cross-section parameters in the window that opens. Select the column material from the “Material” drop-down list in the construction properties window. Determine the location of the column using the 3D locator.

To add a beam to the building model, click the “Beam” icon on the “Tools” panel. This will display the “Beam” tool properties panel in the properties panel area. A locator for entering points will appear in the graphics window.

 Use the drop-down lists in the “Construction Properties” window to select the material and modelling layer. To select the beam cross-section, use the “Cross-section” button. Then, using the locator, enter the points defining the beam axis line in the graphic window (Fig. 3.2). In the “Beam” tool properties panel, you can set the beam location level to be fixed relative to the bottom or top of the floor.

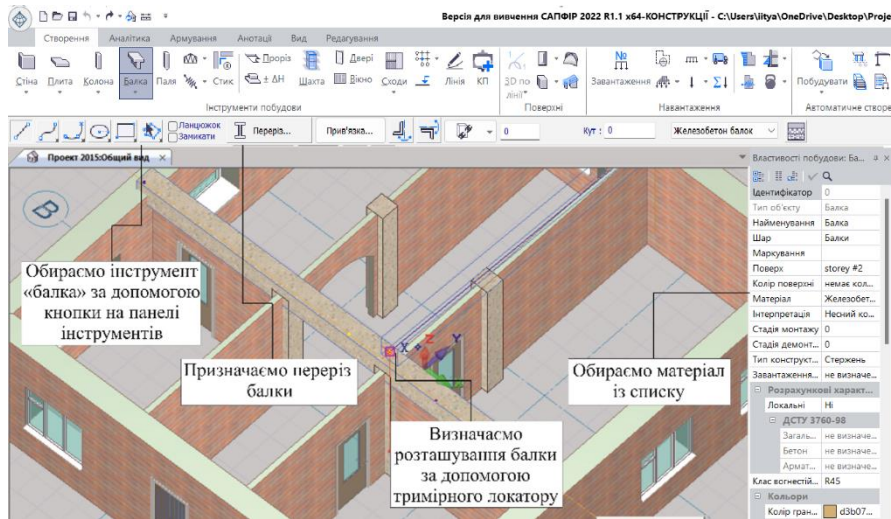
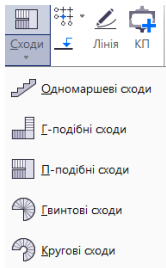


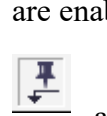
Fig. 2.2 Creating a beam model



Select the “**Stairs**” tool and specify the type of stairs. Set the geometric parameters of the staircase. Set the dimensions of the steps and handrails, specify the colour of the staircase structures and determine the location of the stairs on the floor plan. In the calculation and graphic work, it is permissible not to take the staircase into account in the calculation scheme, so in the “**Construction Properties**” window,



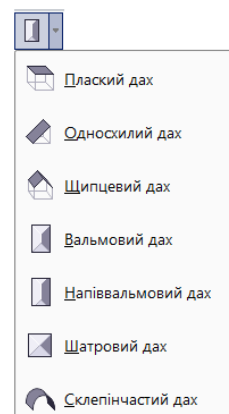
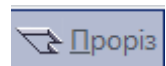
opposite the “**Interpretation**” column, set “**Ignore**”.



Use the “**Slab**” button to call up the corresponding tool. Specify the thickness of the slab. Specify the loads acting on the slab, select the material from the list in the construction properties window. Then use the 3D locator to enter the points of the slab contour. While entering points, you can switch the construction mode to alternate between straight and curved sections of the contour. The “**Chain**” and “**Close**” modes for constructing the slab contour are enabled by default and cannot be disabled, as this is a closed contour object.




If the “**Fix level**” button is enabled when creating a floor slab, regardless of the height at which the contour points are entered, the slab is constructed at the level specified by the number in the editing window, depending on the bottom or top of the floor. If you disable level fixing (press the “**Fix Level**” button), the elevation of the upper plane of the floor slab is determined by the current position of the three-dimensional locator. The locator, moving in space, sets the level of the floor. Use the Shift + Z key combination to enable vertical movement. To adjust the height reference of the floor, use the controls in the “**Level**” group.




To create a hole in the slab, click the “**Slot**” icon on the “**Tools**” panel. This will display the “**Slot**” tool properties panel in the properties panel area. A cursor will appear in the graphics window, allowing you to select objects. Use the cursor to select the object in which you want to make a hole. The locator will switch to point entry mode, and the construction plane will be oriented parallel to the specified object plane. Use the 3D locator to enter points and form the outline of the hole.

To model a roof, select the menu item “**Create**” / “**Roof**” or click the “**Roof**” icon on the “**Tools**” panel. This will display the “**Roof**” tool properties

panel in the properties panel area. In the graphics window, the locator will switch to point entry mode. To create a roof model, select the type, set the slope angle, thickness, overhang size, height reference relative to the floor level (“**Level**” group), material, and modelling layer. Then, using the 3D locator, enter the points that define the roof base contour without taking into account the overhang in the graphics window. In the calculation and graphic work, it is permissible not to take into account the load from the roof in the calculation scheme, so in the “**Construction Properties**” window, opposite the “**Interpretatio**” column, set “**Ignore**”. Roof modelling should be approached after creating a new floor.

 To build the load-bearing structures of the second floor and attic, you will need to create a new floor. Select the “**Create/Copy Floor**” command. This will initiate a **Поверх** dialogue box titled “**Create New Floor**”. Specify the number of floors, their height and location. Check the ‘Copy elements’ function and use the filter to select the elements to be copied. After confirming your selection, remove the unnecessary elements from the architectural model of the building.

After constructing the walls of the attic room for trimming the latter with the roof, open the “**Edit**” menu tab, select the “**Wall**” and “**Roof**” elements between which the trimming is to be performed. Specify the wall and roof in any order. The second object should be specified by pressing and holding the Shift key. The selected objects are highlighted in colour.

 Wall trimming under the roof is usually performed for walls. However, this operation can also be performed for objects of the “**Roof**” type. In this case, trimming is performed for the “**Roof**” object specified first. Trimming can be performed if the objects intersect in height. If the wall is lower, it is not extended to the height of the roof. Therefore, to avoid gaps, set the initial height of the wall with a margin.

Підрізка

Working with the calculation model. The “**Analytics**” tab provides tools for creating and editing an analytical model designed for further calculation using the finite element method and analysis of the stress-strain state of the structure.

This mode provides specific tools for working with the calculation model that are not available on the toolbar of the “**Create**” tab and are designed for creating and editing objects of the corresponding types.

If analytical models have not yet been created in the existing project, the “**Create New Calculation Model**” dialogue box will appear when you try to switch to the calculation model. When you click “**OK**” in this dialogue box, a new calculation model will be created in the project under the “**Analytics**” section. Several analytical models can be created, each of which will characterise a certain aspect of the structure's performance, so the calculation models are named “**Variant 1**”, “**Variant 2**”, etc. You can create a new variant of the analytical model of the building at any time, for example, by selecting “**Create**” / “**Calculation model**” from the menu bar or by using the “**Update**” / “**Create new calculation model**” button in the “**Calculation model: creation**” tool block.

The analytical model is a combination of structures (in the form of rods and plates) and loads of the building's design scheme. In the work, the following buttons are used to apply the finite element mesh to the rods and plates:

The “**Triangulation Settings**” command opens a dialogue box that allows you to configure the parameters of the generated finite element mesh. SAPFIR has three finite element mesh generators: one triangular and two quadrangular. Both quadrilateral generators can generate triangular elements in addition to quadrilateral mesh elements. The difference between the two quadrilateral generators is how they behave near supports and in the middle of the span. Meshes generated by a quadrilateral generator are regular in span areas, while meshes generated as adaptive quadrilateral support contours.

The main parameter for generating plate meshes is “**Step**”, which determines the maximum length of the edge of a triangle or quadrangle in the mesh. In addition to this parameter, the “**Add new points**” parameter is useful, which, when set to “**No**”, allows you to create a finite element mesh by combining only existing reference points and segments. There is only one parameter for generating bar meshes: “**Bar division by length**”, which determines the maximum length of the finite element into which the bar is divided. The command “**Triangulate selected bars and plates**” performs triangulation, i.e. creates a finite element on the element.

After performing the above operations, the calculation scheme should be checked and opened in the LIRA-CAD PC using the corresponding buttons in the “**Calculation in LIRA-CAD**” block of the ribbon menu. This will open the main module of the software package.

The software package provides for the automatic generation of calculated force combinations in accordance with the regulatory documents applicable to the design of construction projects. Parallel calculations should be performed in accordance with Ukrainian and European standards. The results of both calculations should be summarised in a table for visual comparison.

Table 2

**Example of a comparative summary table of calculations and results
(State Building Standards vs. Eurocode 2)**

Calculation stage	Designation / Formula	DBN V.2.6-98	EN 1992-1-1	Difference, %
Load calculation				
Design load	q_d / q_{Ed}	24.73 кН/м	30.56 кН/м	+23.6 %
Calculated bending moment	M_d / M_{Ed}	111.26 кНм	137.52 кНм	+23.6 %
Strength of materials				
Calculated strength of concrete	f_{cd}	14.5 МПа	11.33 МПа	- 21.9%

Calculation stage	Designation / Formula	DBN V.2.6-98	EN 1992-1-1	Difference, %
Calculated strength of steel	f_{yd}	435.0 МПа	434.8 МПа	- 0.05%
Required reinforcement area	$A_{s,req}$	606.8 mm²	788.1 mm²	+29.9 %

The comparative analysis structure used in this case study (Load coefficients → Calculated forces → Material strength → Resistance calculation → Final result) is a universal tool. This approach can be used to compare any two building codes for any type of structural element (columns, slabs, foundations) and for any type of load (shear force, torsional moment, axial force).

REFERENCES

Used

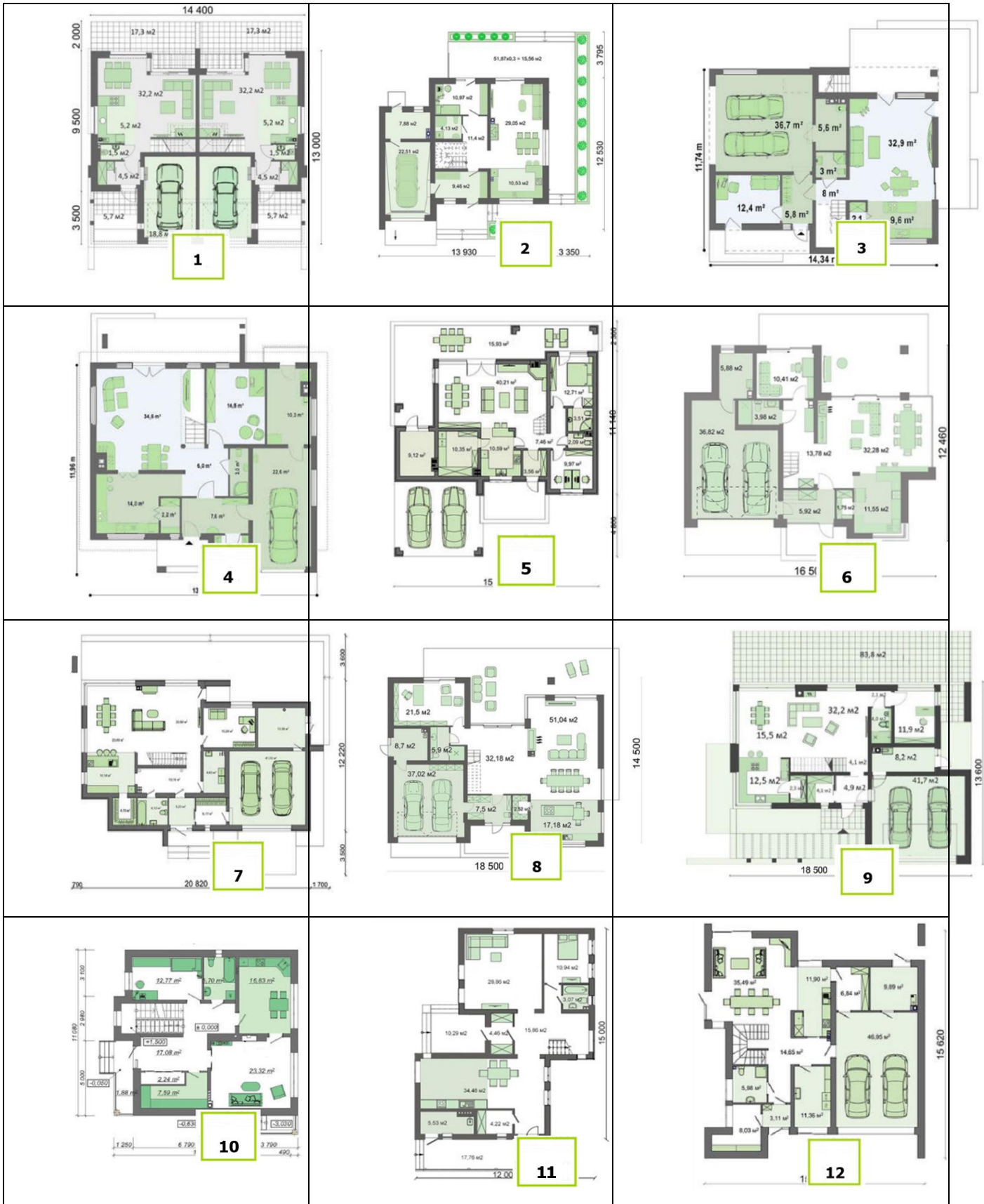
1. State Building Standards B.2.6-98:2009. Concrete and reinforced concrete structures. Basic provisions.
2. State Standards of Ukraine-Н Б EN 1992-1-1:2010. Eurocode 2: Design of reinforced concrete structures. Part 1-1. General rules and rules for structures (EN 1992-1-1:2004, IDT).
3. State Building Standards B.1.2-2:2006. System for ensuring the reliability and safety of construction projects. Loads and impacts. Design standards.
4. State Building Standards B.1.2-14:2018. System for ensuring the reliability and safety of construction projects. General principles for ensuring the reliability and structural safety of buildings and structures.

Recommended for independent study by students

1. Klymenko F.Ye., Barabash V.M., Storozhenko L.I. Zalizobetonni ta kam'iani konstruktsii: Pidruchnyk. – Lviv: Vydavnytstvo Natsionalnoho universytetu "Lvivska politekhnika ", 2009.
2. Bambura A.M., Valovoi D.V. ta in. Proektuvannia zalizobetonnykh konstruktsii za yevrokodamy. Posibnyk z proektuvannia do DSTU Н Б EN 1992-1-1:2010. – Kyiv: DP "NDIBK ", 2012.
3. Mosley B., Bungey J., Hulse R. Reinforced Concrete Design to Eurocode 2. – 7th ed. – Palgrave Macmillan, 2012.
4. The Concrete Centre. "How to design concrete structures using Eurocode 2" (серія посібників та прикладів розрахунків).
5. Official training materials and manuals for LIRA-CAD.

APPENDIX

Input Data for Case Study 2



6. CALCULATION OF THE LOAD-BEARING CAPACITY OF FRAMES USING THE LIRA-SAPR SOFTWARE

Surianinov M. H., Soroka M. M., Pushchina N. V.

INTRODUCTION

This case study was developed as a part of “Information modeling in structural mechanics” course of “BIM engineering” educational and research programme.

The aim of the workshop is to learn how to use the LIRA-CAD software to calculate the load-bearing capacity of frames.

A brief overview of the theoretical principles and assumptions, the method for determining the limit load using the direct method with a calculation example is provided.

When teaching the methodology for using the LIRA-CAD software, a step-by-step algorithm of actions for performing the calculation is described.

The appendix contains the initial data, based on which students must independently calculate the load-bearing capacity of frames using the LIRA-CAD software.

1. BASIC INFORMATION AND ASSUMPTIONS

The load-bearing capacity of the frame is calculated based on the following assumptions [1].

1. A flat frame under simple static load is considered.
2. The cross-section of the frame changes to a plastic state only under the effect of bending moments; the effect of longitudinal and transverse forces is not considered.
3. Cross-sections in which the bending moment is less than the ultimate moment operate under Hooke's law.
4. A plastic hinge is formed in the cross-section where the ultimate moment acts.
5. The joint rotation of cross-sections adjacent to the plastic hinge increases indefinitely without any increase in the bending moment.
6. Under the action of the ultimate load, the frame or its parts do not lose stability.

When determining the ultimate moment for a reinforced concrete cross-section, only the tensile reinforcement is taken into account.

Several methods can be used to determine the ultimate load parameter and construct the ultimate moment diagram and plastic failure diagram of the frame.

This practicum describes the direct method [2], which is based on a step-by-step calculation of the frame as the load parameter increases, with a simple hinge installed at each stage in the cross-section where a plastic hinge has formed. The method allows you to consider all stages of the frame's operation: its elastic operation, elastic-plastic operation, and the stage of destruction, but requires multiple static recalculations.

2. METHOD FOR DETERMINING THE ULTIMATE LOAD WITH THE DIRECT METHOD

The algorithm for calculating the load-bearing capacity of the frame using the direct method can be briefly described as follows:

1. Static calculation of the frame with load parameter ΔF .
2. For each of the calculated frame cross-sections, the plasticity condition is recorded

$$|M_j| \leq M_{0j} \quad (j=1,2,\dots,s), \quad (2.1)$$

where M_{0j} – value of the ultimate moment in the cross-section "j"; M_j – moment arising in the cross-section "j" from load action with parameter ΔF ; s – number of calculated frame cross-sections.

The load value ΔF^1 corresponding to the appearance of the first plastic hinge is established.

$$\Delta F^1 = \min \left(\frac{M_{0j}}{|M_j|} \right) \quad (j=1,2,\dots,s), \quad (2.2)$$

The first plastic hinge is formed in the cross-section where condition (2.2) is satisfied.

3. A moment diagram is being constructed M_{rp}^1 , corresponding to the load ΔF^1 .

$$M_{rp,j}^1 = \Delta F^1 M_j, \quad (2.3)$$

4. A simple hinge is installed in the cross-section where the plastic hinge was formed. This eliminates the possibility of a change of the bending moment in this cross-section with a further increase of the load parameter.

5. If the frame or part of it has become geometrically variable, go to step 10.

6. Perform a static calculation of the frame with the hinge installed under the load parameter ΔF .

7. Determine the cross-section where the next plastic hinge is formed and the load parameter increasing ΔF^i that causes its appearance.

To do this, use the following formula

$$|M_{rp,j}^{i-1} + \Delta F^i M_j| \leq M_{0,j}, \text{ або } \begin{cases} +(M_{rp,j}^{i-1} + \Delta F^i M_j) \leq M_{0,j} \\ -(M_{rp,j}^{i-1} + \Delta F^i M_j) \leq M_{0,j} \end{cases} \quad (2.4)$$

Using (2.4), the smallest positive value of ΔF^i and the cross-section where the plastic hinge is formed are determined.

8. A moment diagram M_{rp}^i is constructed, corresponding to the load parameter $F = \sum_{k=1}^i \Delta F^k$;

$$M_{rp,j}^i = M_{rp,j}^{i-1} + \Delta F^i M_j \quad (2.5)$$

9. The calculation is repeated, starting from step 4.

10. The load parameter and moment diagram obtained in step 8 are the limits for this frame.

Example 1. Calculation of a frame using the direct method

Determine the limit load parameter, construct a limit moment diagram and a plastic failure mechanism for the frame shown in Fig. 2.1, a.

The degree of static uncertainty of the frame is $n=1$. To form a complete plastic failure mechanism, two plastic hinges must occur.

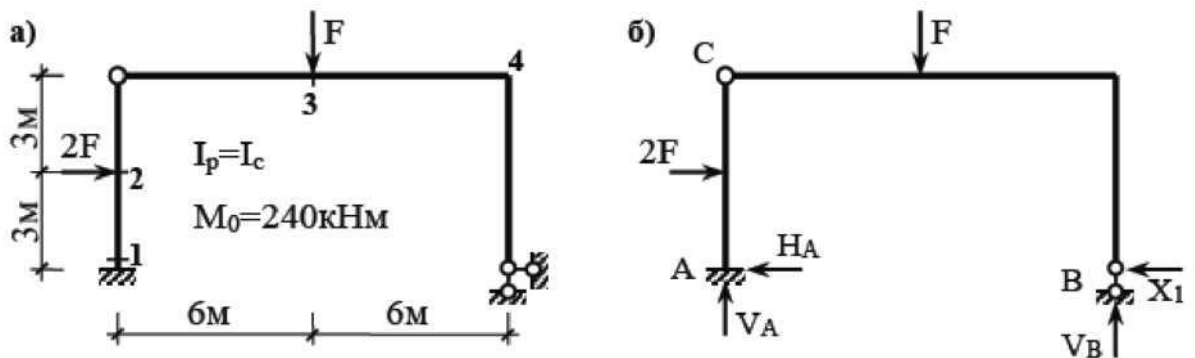


Fig. 2.1 Initial data for calculation (a) and the basic system of the force method (b)

First stage of calculation

The basic system of the force method is shown in Fig. 2.1b, unit and load moment diagrams are shown in Fig. 2.2a.

Canonical equation of the force method:

$$\delta_{11} X_1 + \Delta_{1f} = 0$$

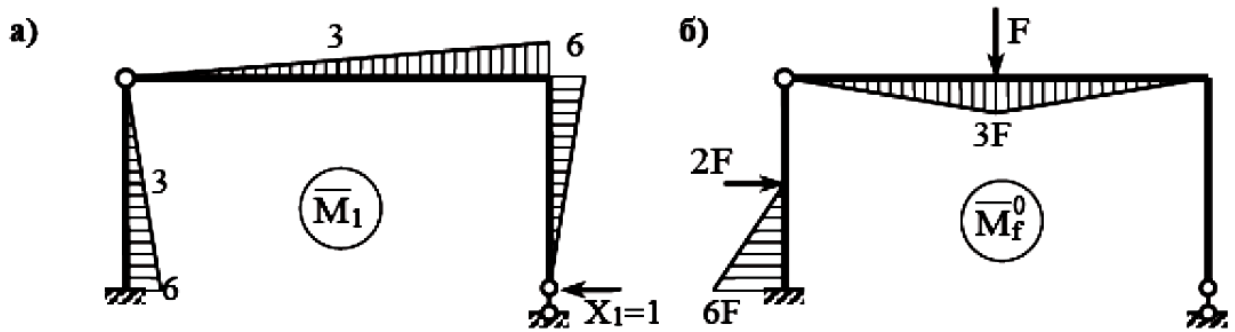
Calculate unit and load displacement:

$$\delta_{11} = \sum \int_0^l \frac{\bar{M}_1^2}{EI} dx = \frac{1}{EI} \left[\left(\frac{1}{2} \cdot 6 \cdot 6 \cdot \frac{2}{3} \cdot 6 \right) \cdot 2 + \frac{1}{2} \cdot 6 \cdot 12 \cdot \frac{2}{3} \cdot 6 \right] = \frac{288}{EI};$$

$$\begin{aligned} \Delta_{1f} &= \sum \int_0^l \frac{\bar{M}_1 M_f^0}{EI} dx = \\ &= -\frac{1}{EI} \left[\frac{1}{2} \cdot 6F \cdot 3 \cdot \left(3 + \frac{2}{3} \cdot 3 \right) + \frac{1}{2} \cdot 3F \cdot 6 \cdot \frac{2}{3} \cdot 3 + \frac{1}{2} \cdot 3F \cdot 6 \cdot \left(3 + \frac{1}{3} \cdot 3 \right) \right] = -\frac{99F}{EI}. \end{aligned}$$

Fig.

2.2 Single (a) and load (b) moment diagrams



We determine the unknown of the Force Method:

$$X_1 = -\frac{\Delta_{1f}}{\delta_{11}} = \frac{99F}{288} = 0,34375F.$$

$$M^1 = \bar{M}_1 X_1 + M_f^0$$

We construct a “flexible” moment diagram for the first stage (Fig. 2.3).

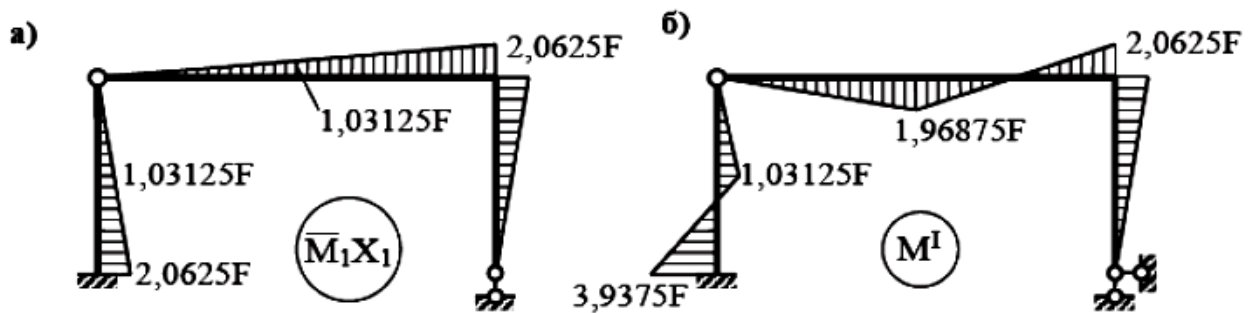


Fig. 2.3 “Flexible” moment diagram of the first stage

We determine the limit load parameter of the first stage. The maximum bending moment occurs in section 1, which means that the first plastic hinge is formed in the same cross-section. We equate the bending moment in section 1 to the ultimate moment and determine the ultimate load of the first stage F^1 .

$$3,9375F^1 = M_0;$$

$$F^1 = \frac{M_0}{M_1^1} = \frac{240}{3,9375} = 60,95238 \text{ kH.}$$

$$M_{sp}^1 = M^1 F^1.$$

We construct the limit moment diagram for the first stage of calculation (Fig. 2.4).

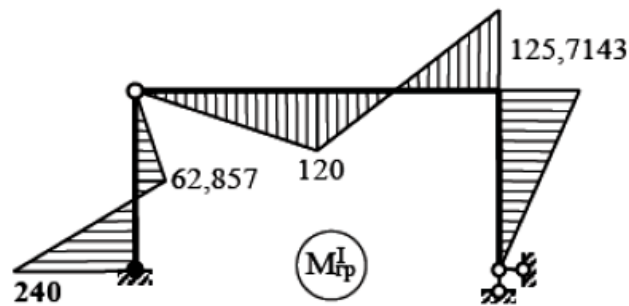


Fig. 2.4 Limit moment diagram for the first state

Second stage of calculation

We install a simple hinge in cross-section 1, where a plastic hinge has formed, and construct a “flexible” moment diagram from the loading of the frame with load increasing by ΔF (Fig. 2.5). After installing the simple hinge, the frame became statically determinate.

Support reactions:

$$\begin{aligned} \sum m_C^{res} &= 6H_A - 3 \cdot 2\Delta F = 0; \\ H_A &= \Delta F. \\ \sum X &= 2\Delta F - \Delta F - H_B = 0; \\ H_B &= \Delta F. \\ \sum m_A &= 3 \cdot 2\Delta F + 6\Delta F - 12V_B = 0; \\ V_B &= \Delta F. \end{aligned}$$

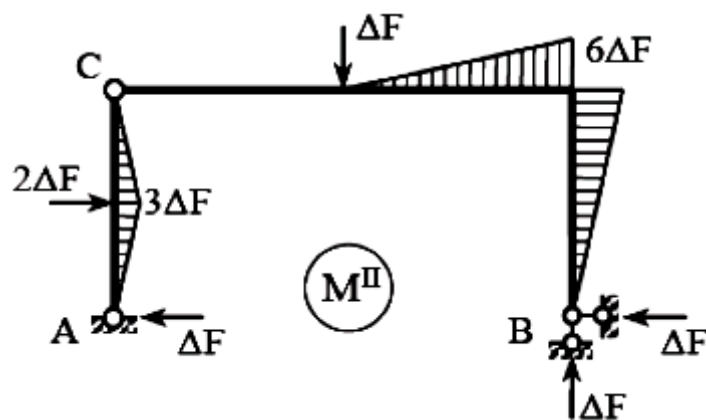


Fig. 2.5 “Flexible” diagram of moments of the second stage

Analysing the moment diagrams M_{rp}^I and M^II , we note that the largest moments occur in section 4. Based on this, we determine the increasing load parameter at which the second plastic hinge is formed:

$$6\Delta F + 125,7143 = 240 \Rightarrow \Delta F = 19,04762\kappa H.$$

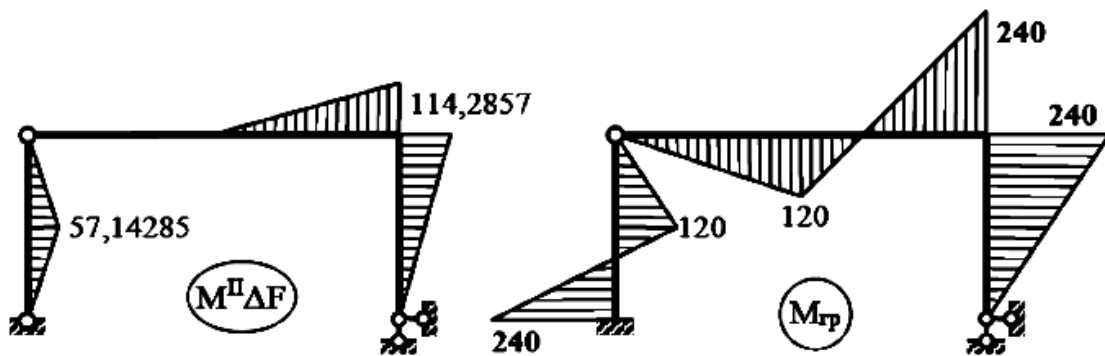
The maximum load parameter equals to:

$$F_0 = F^I + \Delta F = 80\kappa H.$$

We construct a limit diagram of bending moments (Fig. 2.6).

$$M_{ep} = M^II \Delta F + M_{ep}^I$$

Fig. 2.6 Limit diagram of bending moments



We plot the plastic mechanism of frame destruction (Fig. 2.7) and construct a remaining bending moment diagram (Fig. 2.8), which arises during the plastic deformation of the frame and does not disappear after the external load is removed.

$$M_{\text{ост}} = M_{ep} - M^I F_0.$$

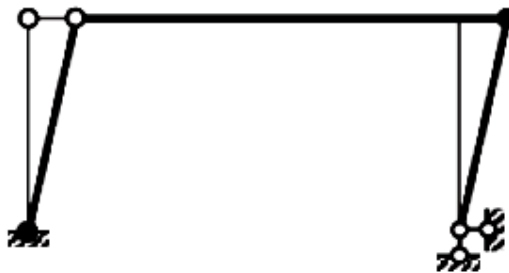


Fig. 2.7 Plastic mechanism of frame destruction

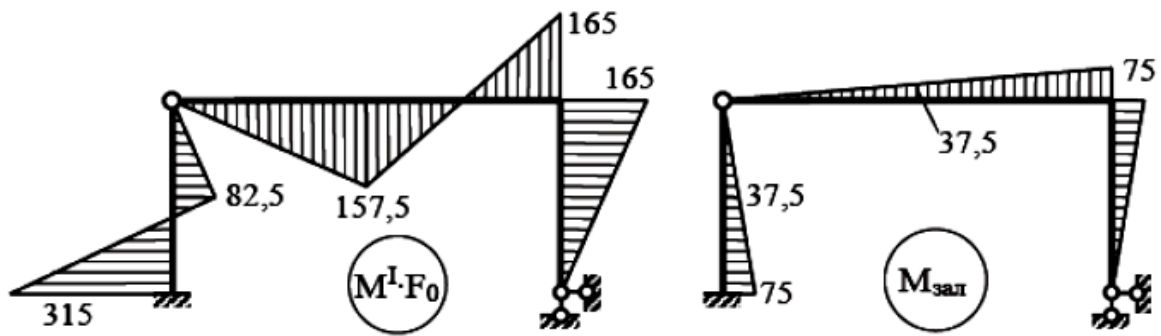


Fig. 2.8 Residual moment diagram

Since M_{3an} diagram is created after the load is removed, there should be no breaks in the places where concentrated forces were applied.

3. METHODOLOGY FOR DETERMINING THE ULTIMATE LOAD USING LIRA-CAD SOFTWARE

In the LIRA-CAD software package [3], the limit forces in the cross-sections of the structure are controlled using a special two-node element CE-255, which allows modelling both linear and inclined compliance relative to local coordinate axes. The values of the limit forces, after which yielding will start in a given element, are entered in the element stiffness parameters. Such elements should be entered into the calculation model in those places where plastic hinges are expected to form.

An iterative process based on a simple step method is used to find the limit load. The accuracy of the calculation depends on the number of iterations and the load step.

Example 2. Calculation of a frame using the LIRA-CAD software

Perform the calculation of the frame shown in Fig. 2.1.a.

Some preparatory work is required before performing the calculation. Determine the cross-sections of the frame where plastic hinges may form. These are the places where rigid joints are installed, the points of application of concentrated forces, and the points of rigid connection of the frame members. In such places, it is necessary to create two nodes located at a distance of 1 cm from each other (Fig. 3.1).

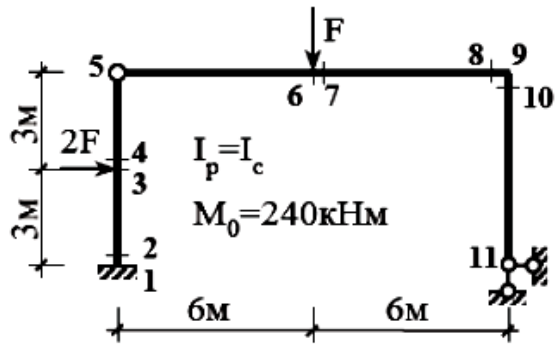


Fig. 3.1 Numbering of frame nodes

Having assigned the beginning of the coordinates to node 1, we form a table of frame node coordinates (Table 3.1).

Table 3.1

Frame node coordinates

№ node	Coordinates of node (m)	
	x	z
1	0,00	0,00
2	0,00	0,01
3	0,00	3,00
4	0,00	3,01
5	0,00	6,00
6	6,00	6,00
7	6,01	6,00
8	11,99	6,00
9	12,00	6,00
10	12,00	5,99
11	12,00	0,00

After starting LIRA-CAD PC, select the type of diagram – 2 (Fig. 3.2), enter the name of the task and press button .

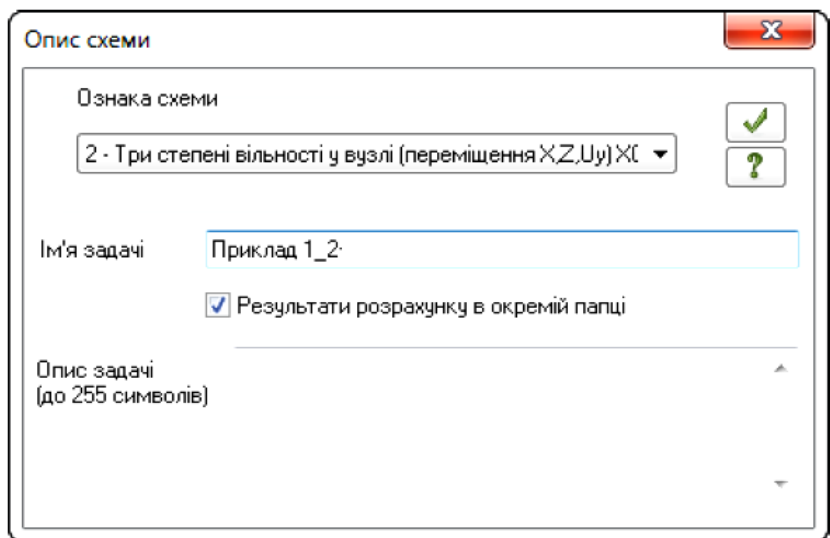


Fig. 3.2 Scheme type selection window

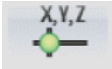
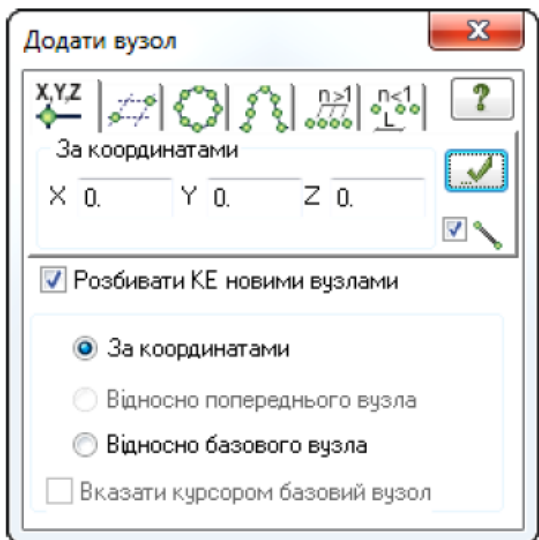

On the “Creation” panel (“Creation and Editing” tab), click the button  and enter the coordinates of the nodes from Table 3.1 (Fig. 2.11) in sequence. After entering the coordinates of the next node, click the button



Fig. 3.3 Window for entering node coordinates



On the “Create” panel, click the button  and connect nodes 1-5, 5-9, and 9-11 with the rods. This creates rods of "normal" length and rods 1 cm long in the frame.






In node 1, we set up rigid fastening. To do this, select node 1 (button on the bottom panel), press the button – “Joints”,  select joints along X, Z, UY and  press the button  (Fig. 3.4). In node  11, set linear joints along X and Z. To do this, select no  in the “Joints in the nodes” window, de-select the UY joint and press the button .



Fig. 3.4 Window for entering supporting joints

Install the hinge in node 5. The hinge can be installed either in the post or in the crossbar. Select the crossbar rod, go to the **“Rods”** tab, click the button, place a UY mark in the first node (Fig. 3.5) and click the button.

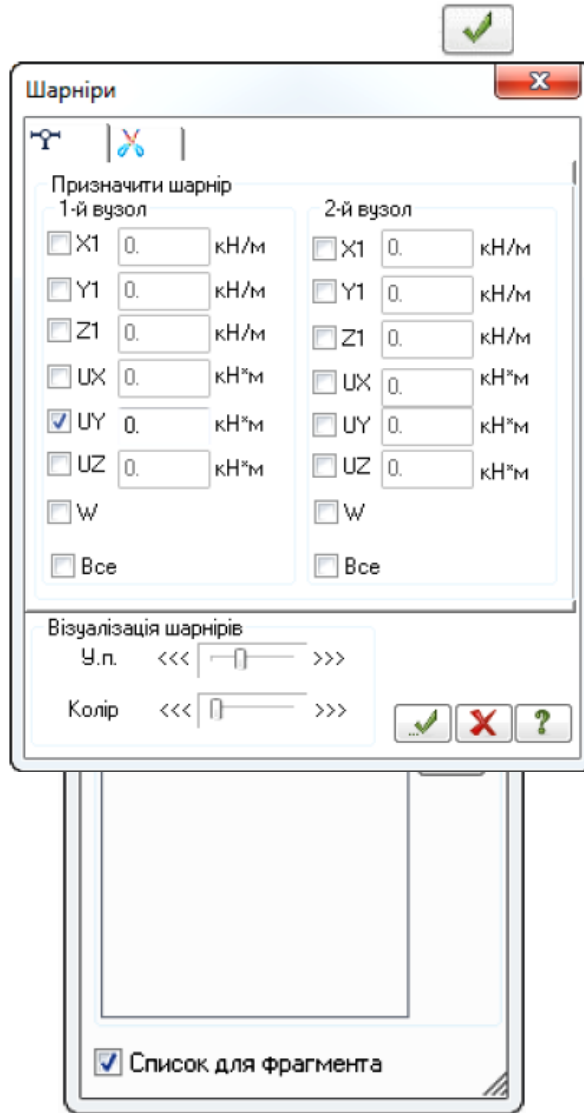

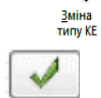


Fig. 3.5 Window for setting up hinges

Select all frame rods (button on the bottom panel), go to the **“Advanced editing”** tab and click the button. In the  window (Fig. 3.6), select **“Type 2 – CE flat frame”** and click

the button.



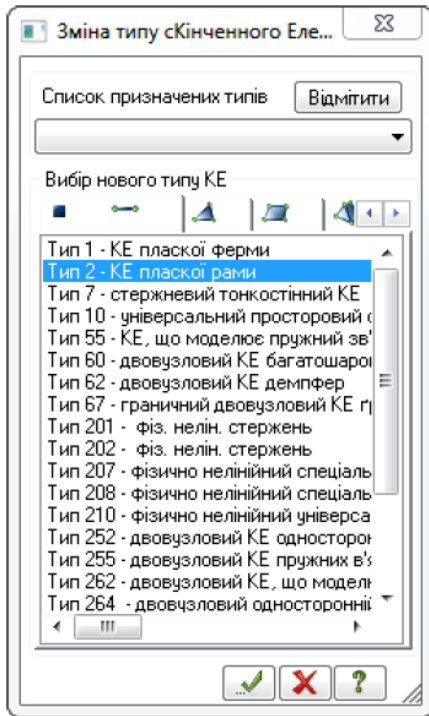

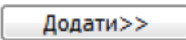





Fig. 3.6 Window for changing the type of finite element

Go to the “Create and Edit” tab and click the button  Жорсткості on the “**Rigidity and Binding**” panel. In the “**Rigidity and Materials**” window (Fig. 3.7), click the button , go to the EF tab and double-click on  KE 2  чисельне .

In the window for entering numerical values for stiffness (Fig. 3.8), enter

$$EF = 1e8кН; EI = 1e6кН \cdot м^2$$

and press the button  . These random stiffness values are entered only so that the frame can withstand external loads, since in this case ($I_p=I_c$) the stiffness values do not affect the calculation of the limit load. Select the frame members, except for those with a length of 1 cm, and click the button in “**Stiffness and Materials**” window.

By selecting these frame members,  they will be assigned the entered stiffness.

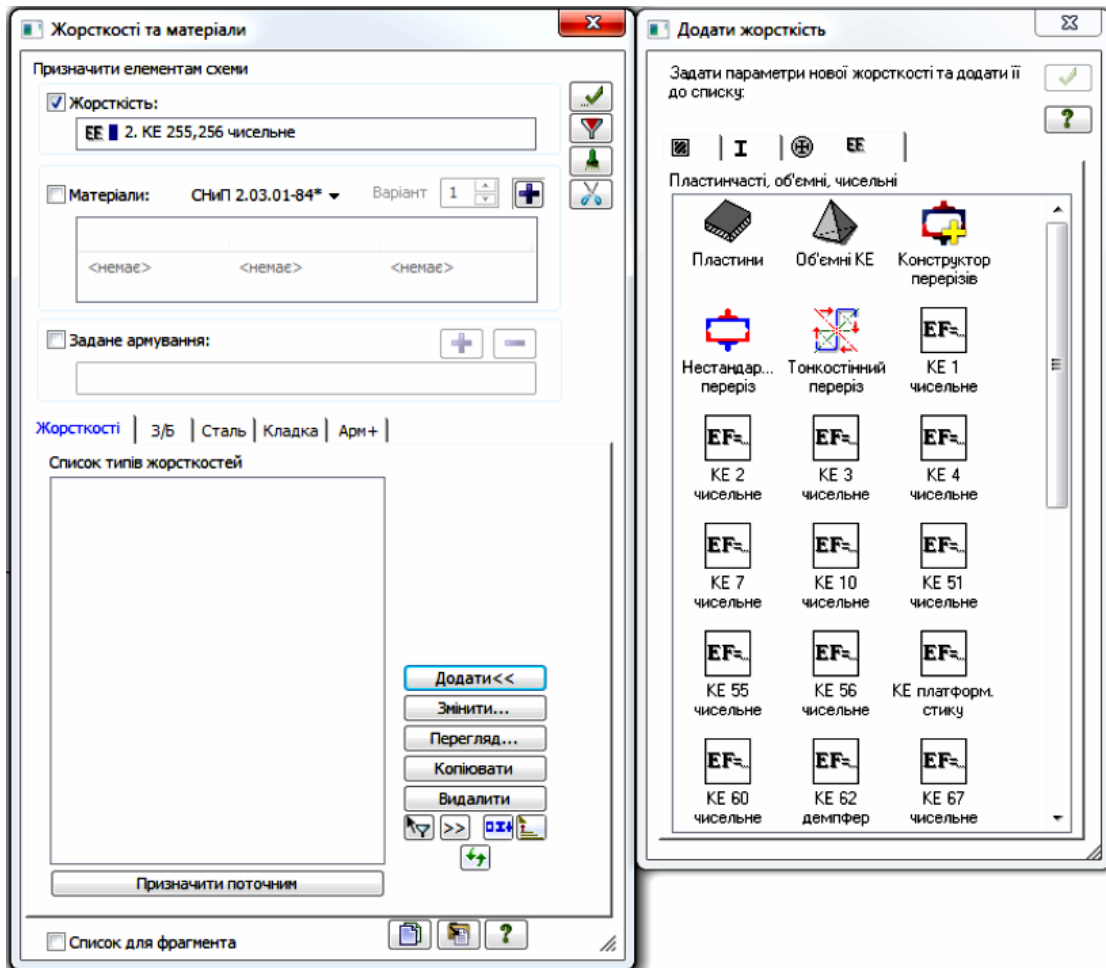




Fig. 3.7 Window for entering finite element stiffnesses

The core CE, they are 1 cm long, need to be changed to CE255. To select them on the bottom panel, press the button – “Polyfilter”. In the  “Polyfilter” window, go to the “Filter for elements” tab, check the “By stiffness type”  box, select “No” and press the button (Fig. 3.9). This will select the bars that have not been assigned stiffness, i.e. bars with a length of 1 cm. Next, go to the Advanced Editing tab and click the button. In the window (Fig. 3.6), select “Type 255 – Two-node CE elastic joints with boundary forces” and click the button.



Зміна типу KE

KE 2 чисельне X

EF	<input type="text" value="1e8"/>	кН
Ely	<input type="text" value="1e6"/>	кН*м ²
Elz	<input type="text" value="0"/>	кН*м ²
Glk	<input type="text" value="0"/>	кН*м ²
Y1	<input type="text" value="0"/>	см
Y2	<input type="text" value="0"/>	см
Z1	<input type="text" value="0"/>	см
Z2	<input type="text" value="0"/>	см
Ru_Y	<input type="text" value="0"/>	см
Ru_Z	<input type="text" value="0"/>	см
Elw	<input type="text" value="0"/>	кН*м ⁴
q	<input type="text" value="0"/>	кН/м

Врахування

GF GFy кН

 GFz кН

Коментар

Колір

Fig. 3.8 Window for entering stiffness values

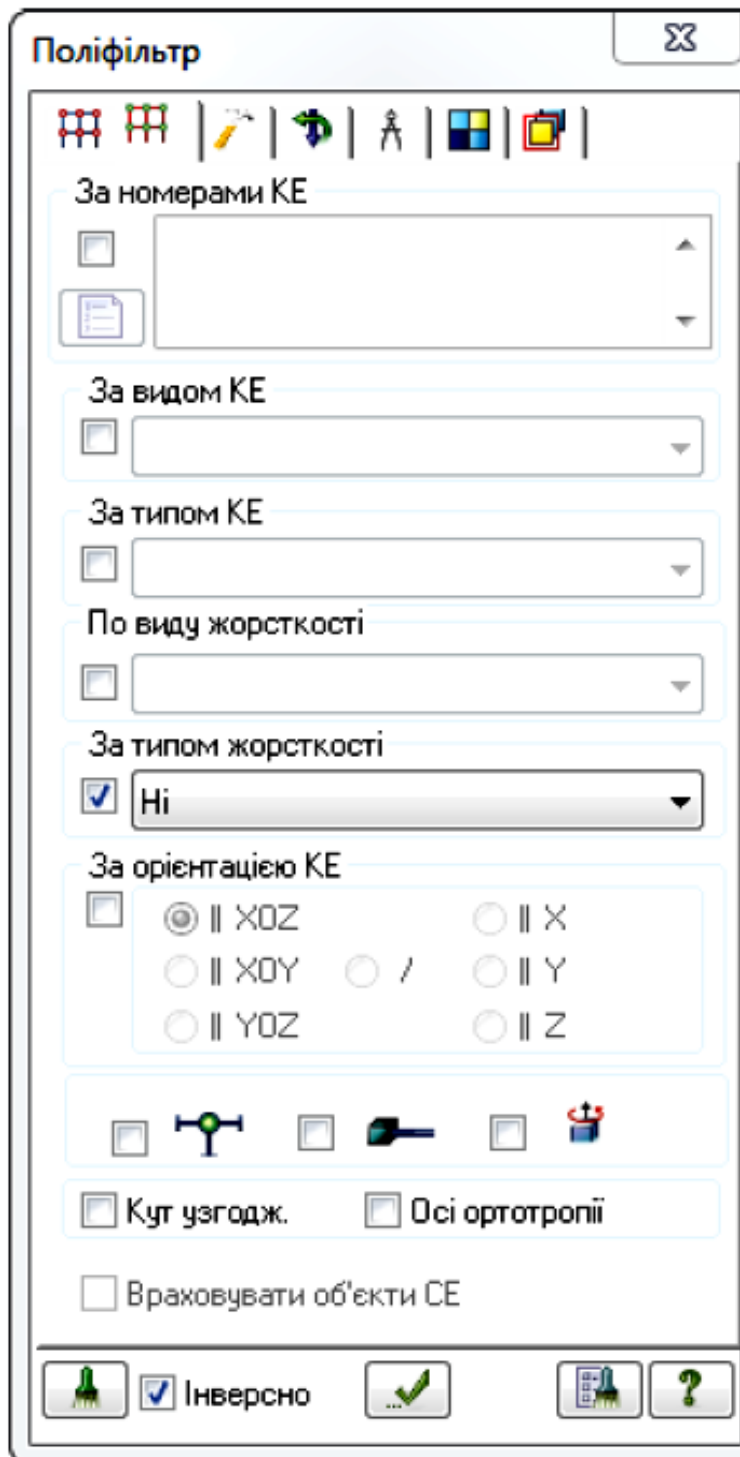


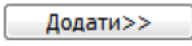


Fig. 3.9 Filter window for the elements

Select CE 255. To do this, click the “Polyfilter” button. In the  “Polyfilter” window, go to the “Filter for elements” tab, check “By CE type” box, select “Type 255 – two-node...” and click the button.



Go to the “Creation and Editing” tab and click the button  Жорсткості on the “Stiffness and Connections” panel. In the “Stiffness and Materials” window, click the button , go



KE 255,256

to the *EF* tab and double-click on *чисельне*. Enter the stiffness CE 255 (Fig. 3.10) and click the button “The running stiffness” *R* and the ultimate value of the longitudinal forces are entered randomly, but it is necessary to ensure that the ultimate values of the longitudinal forces are significantly greater than the longitudinal forces arising in the members from the load. In the “Stiffnesses and materials” window, click the button

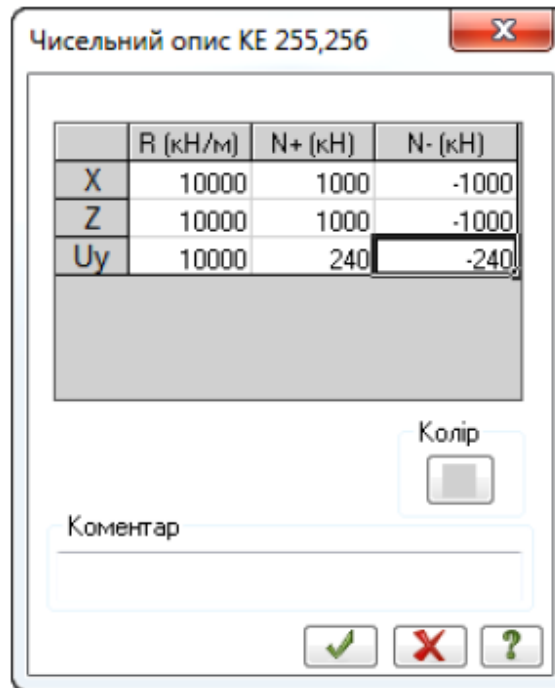



Fig. 3.10 Window for entering stiffnesses CE 255

Go to “Creation and Editing” tab and click “Load on Nodes” button on “Load” panel. To enter a horizontal concentrated force in node 3, select the direction of force X,

click the button , enter the force value -2 (Fig. 3.11) and click the button. Select node 3 and click the button in the “Load Assignment” window.



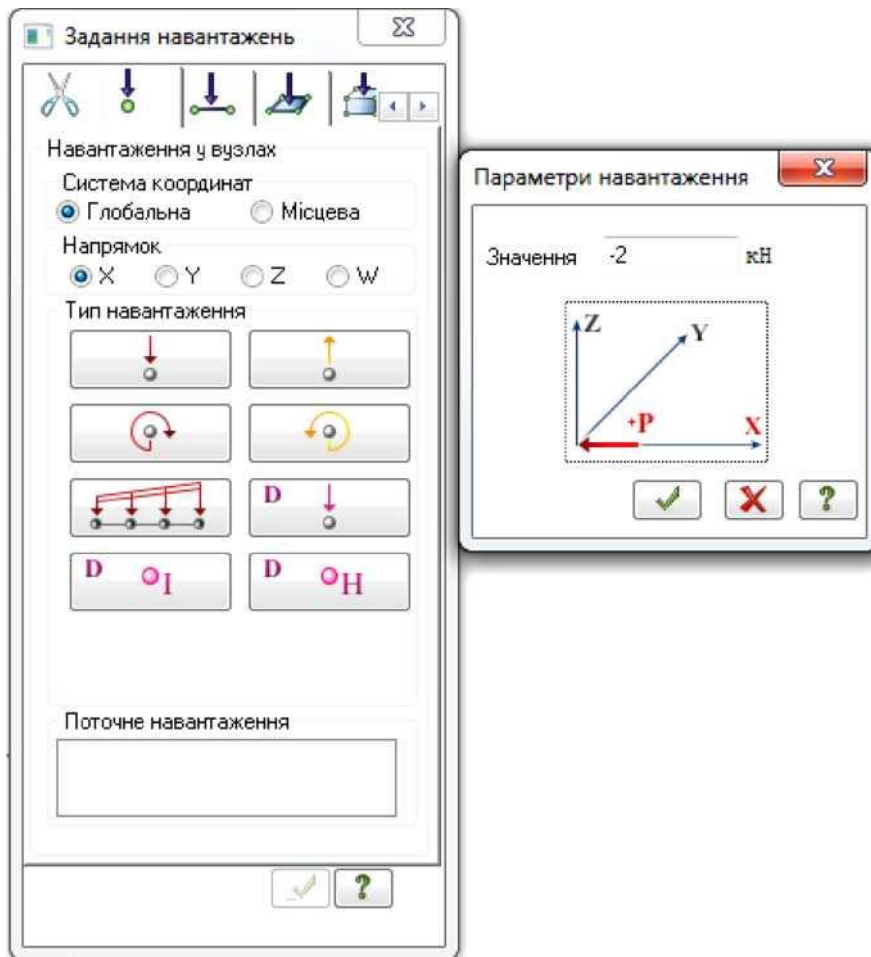



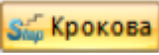




Fig. 3.11. Load input window

To apply a vertical force to node 6, select the Z force direction, press button , enter the force value – 1, and press the button Select node 6 and press  the button  in “Load Assignment” window.

Next, go to “Calculation” tab, select the type of nonlinear procedure  in “Nonlinearity” panel. In “Simulation of nonlinear loads on structures” window, click the button  to add a nonlinear load, enter the load number – 1, enter the number of steps – 500, select and enter the total coefficient 200, which, in this case, means the maximum load value that can be achieved during the calculation process (Fig. 3.12). 

Since the value of the ultimate load is not known in advance, we enter a value that is known to exceed the ultimate load as the total coefficient. Since the programme does not stop the calculation when the limit load is reached, when the frame turns into a mechanism, but continues the calculation until the specified load is reached, it is necessary to save the intermediate results, the analysis of which will allow determining the limit load. To do this, in the “Output intermediate results” menu, set “Output all”. We set the checkbox “Calculate step by step, even if there are no step elements” and press “Confirm” button.



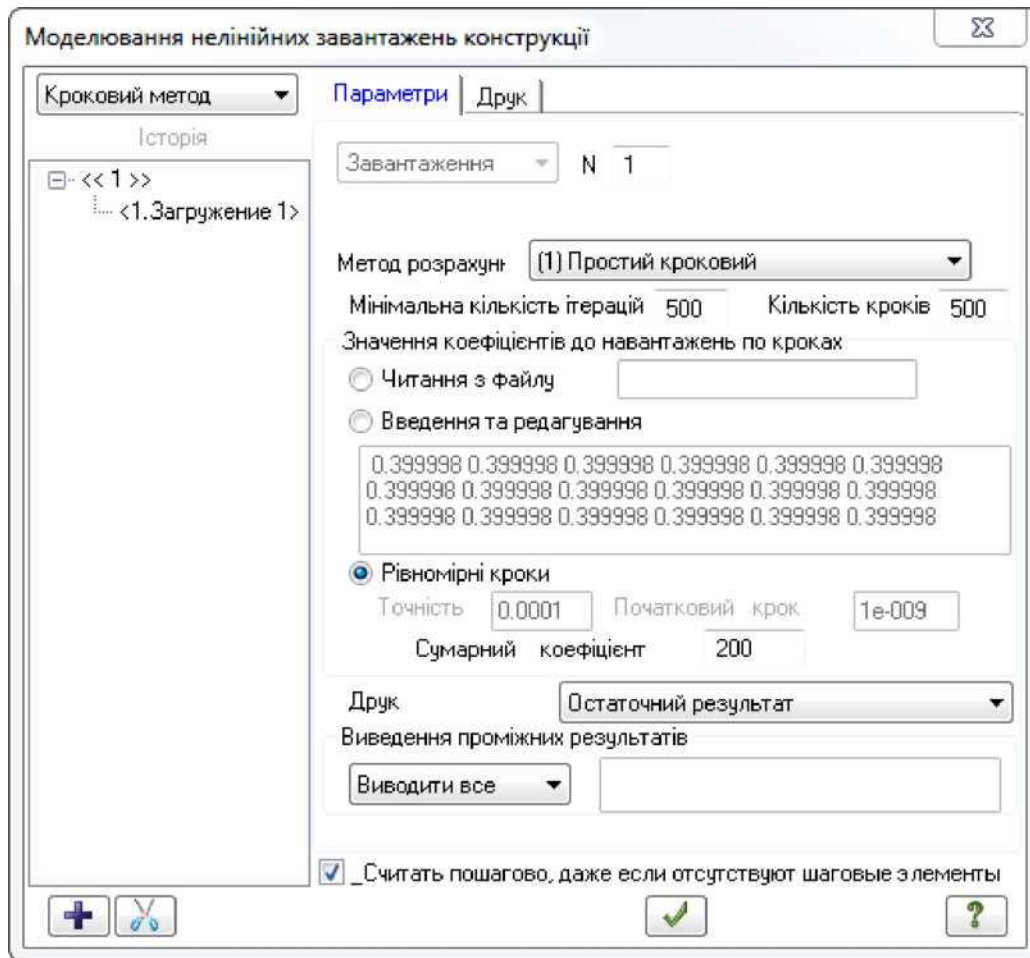




Fig. 3.12 Window for modelling nonlinear loads

On the “**Calculation**” panel or on the quick access panel, click the button  – “**Perform full calculation**” and wait for the calculation to complete. If the “**Total coefficient**” is set incorrectly (Fig. 3.12), the calculation may take a long time because, after reaching the ultimate load, the programme will attempt to calculate a geometrically variable system. To save time, you can stop the calculation and find the last load in the log at which the iterative process ended successfully. This load, increased by the load step value, must be entered as the “**Total coefficient**” (Fig. 3.12) and the calculation must be performed. After completing the calculation, go to the “**Analysis**” tab and display the bending moment diagram (button  on the “**Forces in members**” panel).

Next, open the menu on the bottom panel (Fig. 3.13), where you select the load, which is indicated as a percentage of the “**Total coefficient**”.



Fig. 3.13 Selection of intermediate calculation results

The bending moment diagram will change on the screen according to the selected load. After determining the maximum load at which the iterative process ended successfully from the solution report, select the corresponding load percentage in the “**Form number (component, period)**” menu. To refine the result, select adjacent (higher) load percentage values and ensure that the bending moments in the frame do not exceed the limit moments. We accept the maximum load at which the bending moments do not exceed the limit moments as the ultimate load. Since the load entered as the “**Total coefficient**” is displayed on the screen, we obtain the value of the ultimate load as a percentage of the “**Total coefficient**”. For this example, the limit load $F_0=200*0,4=80\kappa H$ coincides with the result of the analytical calculation. Plastic hinges are formed in sections 1-2, where $M = 240 \kappa H_M$, and in sections 8-10, where $M = 239\kappa H_M$ (Fig. 3.14).

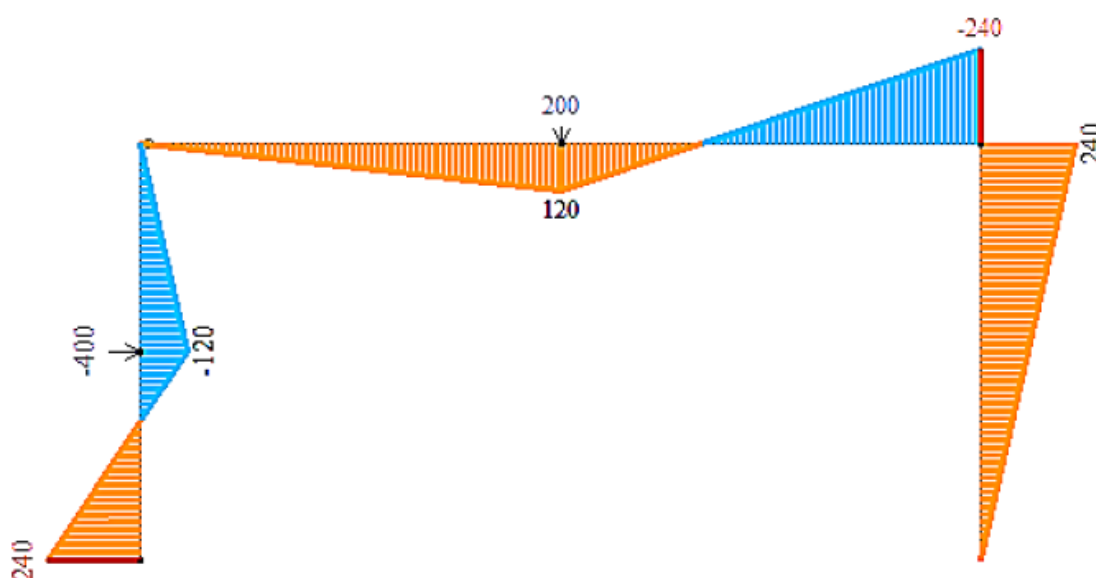


Fig. 3.14 Diagram of bending moments at the ultimate limit state (kNm)

The plastic failure diagram of the frame is shown in Fig. 3.15.

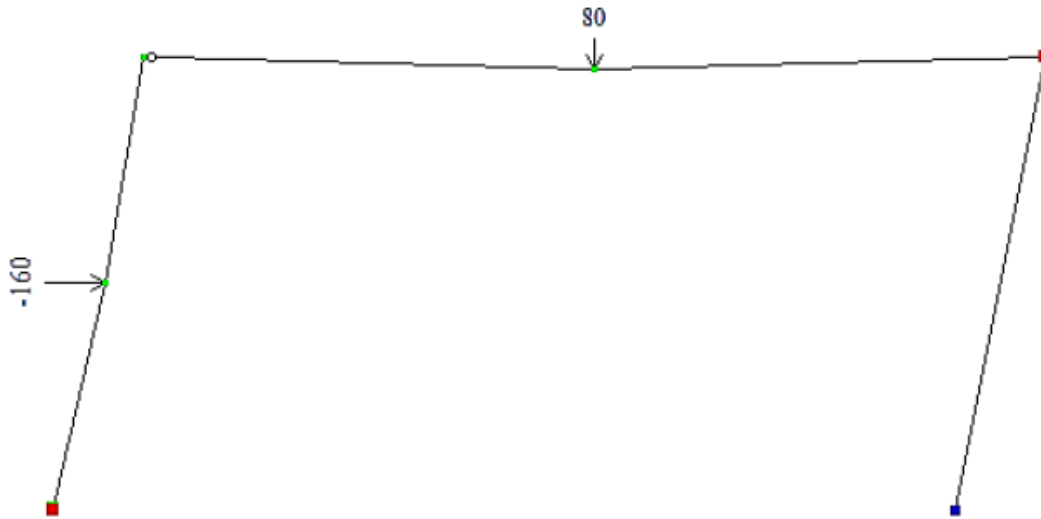


Fig. 3.15 Diagram of plastic failure of the frame

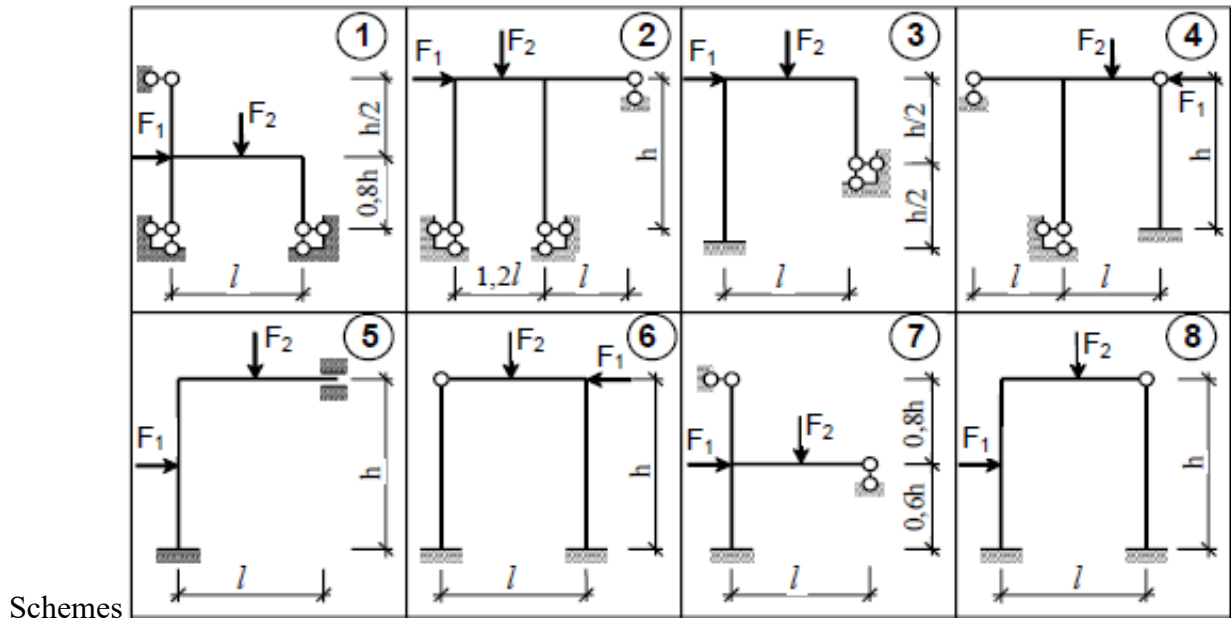
REFERENCES

1. Chykhlyadze E. D. *Budivelna mekhanika: Pidruchnyk*. – Kharkiv: UkrDAZT, 2011. – 320 s., rys. 234, tabl. 14
2. *Neliniina budivelna mekhanika z PK LIRA-SAPR*. M. S. Barabash, M. M. Soroka, M. H. Sur'ianinov. – Kyiv, 2018. – 246 s.
3. Программный комплекс LYRA-SAPR. Рукoвoдствo ползoватeля. Oбучающeгo прымeрy. Vodopianov R. Yu., Tytok V. P., Artamonova A. E., Romashkyna M. A. Pod red. akademyka RAASN Horodetskoho A. S. Электронное издaнe. – 2017. – 535 s.

APPENDIX

Diagrams and Input Data for the Design and Calculation Project

Calculation



Input Data

Table 1

o.	Dimensions					Reinforcement Coefficients (A400C Reinforcement Steel)	
	M)	M)	$bp=bc$ (M)	h_p (M)	h_c (M)	ρ_p	ρ_c
			0,20	0,40	0,30	0,012	0,013
			0,30	0,40	0,30	0,015	0,015
			0,30	0,40	0,35	0,013	0,012
			0,30	0,40	0,35	0,014	0,014
	0		0,40	0,45	0,40	0,016	0,013
	2		0,35	0,45	0,40	0,015	0,014
			0,35	0,40	0,30	0,014	0,017
	2	0	0,40	0,40	0,35	0,013	0,016

Table 2

Concrete Class	F_1/F_2
C16/20	2,0
C20/25	0,4
C20/30	3,0
C30/35	0,8
C32/40	2,0
C16/20	0,5
C20/25	2,5
C25/30	0,25

Concrete Properties

Concrete Class	E_{cm} (kN/m ²)	f_{cd}
C16/20	$2.7 \cdot 10^7$	11500
C20/25	$3.0 \cdot 10^7$	14500
C20/30	$3.2 \cdot 10^7$	17000
C30/35	$3.45 \cdot 10^7$	19500
C32/40	$3.6 \cdot 10^7$	22000

For reinforcement of class A400C, $f_{yd} = 365,000$ kN/m²

7. ANALYSIS AND DESIGN OF STEEL STRUCTURES IN TEKLA STRUCTURES SOFTWARE

Davydov I. I., Chaban V.P.

1. Introduction

The Tekla Structures software in the educational process is used to develop modern professional competencies in the field of design, modelling, and calculation of steel structures using BIM technologies for future construction and design professionals. The aim is not only to study the software itself, but also to provide comprehensive training for a new generation of professionals capable of working with digital models, performing calculations in accordance with national standards and Eurocodes, as well as implementing BIM technologies in the design and operation of buildings.

Thus, the following tasks are set:

- to teach students to work in the integrated environment that combines 3D modelling, automated analysis, and generation of working documentation;
- to demonstrate the advantages of BIM approach compared to traditional design methods (error reduction, automation of routine processes, increased accuracy);
- to familiarise students with national standards (DBN, DSTU) and European standards (EN, Eurocodes);
- to train them to adapt the software to meet the requirements of current regulations;
- to train students to adapt the software to the requirements of current standards;
- to provide opportunities to verify engineering calculations using real structural analysis software packages (SCAD Office and LIRA-SAPR);
- to teach students to compare automated calculation results with manual methods and critically evaluate the adequacy of the computational model;
- to promote understanding of the transition from a BIM model to an analytical model (formalisation of rules, consideration of joint stiffness, and specific aspects of node interpretation);
- to create a foundation for scientific research in the field of digital structural models, design automation, and integration of standards into software packages;
- to develop in students the skills required to work in modern design organisations that use BIM technologies;
- to prepare them to participate in course and diploma projects using Tekla Structures as the main design tool.

After completing the course ‘Analysis and design of steel structures in Tekla Structures software’, students should acquire specific knowledge, skills, and competencies that will enable them to effectively perform design and research tasks in a modern BIM environment. Below is a detailed list of expected outcomes as measurable learning results and the evidence demonstrating their achievement.

The student will acquire theoretical knowledge and:

- know how the BIM platform works and the role of Tekla Structures in integrated design;
- understand the fundamentals of modeling frame and plate systems, as well as the concept of transforming a BIM model into an analytical model.
- understand what stiffness and joint behavior are, as well as their influence on structural analysis and design.
- can navigate national and European standards.

Learning evidence: theoretical test, oral examination, lecture notes.

The student will acquire professional and technical skills and will be able to:

- create an accurate 3D model of a steel structure in Tekla Structures (columns, trusses, joints, support conditions);
- set up and use localized databases of profiles and materials (DSTU/EN);
- develop analytical models considering the actual joint geometry and export them to structural analysis software (*SCAD Office*, *LIRA-SAPR*) or internal modules;
- model joint stiffness and behavior (bolted, welded) and account for them in structural analysis (calculations);
- generate detailed drawings, specifications, and lists of materials with identification codes;

Skills evidence: practical assignments (model + drawings), file export/import, specifications.

The student will develop analytical and research competencies, enabling to:

- compare obtained calculation results using different modeling approaches (rigid, pinned, and semi-rigid joints; simplified and detailed models) and draw conclusions about the influence of assumptions;
- perform validation of the analytical model by comparing results from *Tekla Structures*, manual calculations, and external analysis software;
- carry out sensitivity analysis (influence of joint stiffness, material parameters, and load combinations) on the critical performance indicators of the structure;
- develop and verify simple algorithms or rules for automated transformation from the BIM model to the analytical model.

Competencies evidence: comparative report, research paper, sensitivity analysis -evaluation of the analysis result changes under certain input parameters (materials, loads, joint stiffness, etc.) are varied.

The student will develop design and organizational skills, enabling to:

- organize the workflow within a BIM project, including the use of templates and project parameters;
- prepare a package of working documentation (detailed joint drawings, general layout, specifications) in accordance with client requirements and standards;
- maintain technical documentation for models (comments, annotations, change log);
- collaborate in a team environment: integrate model parts and carry out basic clash detection between model elements.

Skills evidence: group project, change log, clash detection report between model elements.

The student will develop digital (BIM) competencies, enabling to:

- use *Tekla Structures* as a digital representation tool for structural components intended for subsequent monitoring and operation;
- perform data export/import, configure attribute mapping and profile correspondence;
- automate repetitive operations using templates, parametric components, and, if necessary, basic scripts or plugins.

Evidence: integration of structural model data into a 'universal description' of elements (beams, columns, joints, materials, etc.) readable by other software such as ArchiCAD, Revit, LIRA-SAPR, etc.; use of templates; and examples of automation.

The student will develop regulatory competencies, enabling to:

- understand which codes and standards are applied in structural design and analysis;
- correctly apply these standards during calculation and design processes;
- be able to justify the selection of a particular normative approach and identify its source.

Evidence: defense of a course or diploma project with a reasoned selection and application of normative design and analysis schemes, including verified compliance with DBN/DSTU requirements.

Levels of proficiency (recommended classification):

- Basic level: creation of simple models, generation of drawings, basic understanding of design codes.
- Professional level: accurate development of analytical models, integration with structural analysis software, modeling of joints.
- Research/expert level: formalization of BIM-to-analysis rules, validation of models, and structural optimization.

2. Description of a problem

Traditional training of civil engineering specialists is mostly focused on separate stages of design (analysis (calculation), modeling, drafting), which makes it difficult to develop integrated professional competencies. At the same time, in modern construction specialists are required to work in BIM, integrating 3D modeling, structural analysis, and preparation of working documentation.

The use of Tekla Structures in the educational process provides students with systemic understanding of the design process - from creating a digital model of the structure to verifying its compliance with DBN/DSTU and Eurocodes. This approach brings the educational process closer to real engineering practice and enhances the competitiveness of graduates.

Tasks of integrating BIM-oriented learning into the training:

- to teach students to work with digital structural models in *Tekla Structures*;
- to combine 3D modeling, automated analysis, and drawing preparation;
- to adapt analytical models to comply with national standards (DBN, DSTU) and Eurocodes;
- to develop skills for verifying structural analyses in SCAD Office, LIRA-SAPR, and other software packages;
- to teach students to critically evaluate automated analysis results and implement BIM technologies in design practice.

3. Theoretical background

Key concepts of the course topic the case is based on:

- BIM (Building Information Modeling) - a technology for information modeling of buildings, which integrates 3D modeling, structural analysis, and documentation in a single digital model;
- Tekla Structures - a modern BIM platform for the design and analysis of steel structures, as well as for the creation of drawings and material specifications;
- Analytical (calculation) model - a simplified representation of a structure (consisting of beams, plates, and joints) used for engineering calculations.
- stiffness and joint behavior - key parameters that determine the actual performance of a structure (pinned, rigid, and semi-rigid connections);
- DBN/DSTU and Eurocodes (EN) - the regulatory framework for the design and verification of structures;
- integration with structural analysis software (SCAD Office, LIRA-SAPR) - transferring the model from Tekla Structures to specialized programs to verify structural calculations.
- digital competence in BIM – ability to work with 3D structural models, document changes, perform data export/import, and automate design processes.

The course provides a link between the academic curriculum and the real needs of modern construction industry. Prerequisites: *Physics, Theoretical Mechanics, Higher Mathematics, Strength of Materials, Structural Mechanics, Building Materials Science, Steel Structures, Reinforced*

Postrequisites: *Computer Modeling of Steel Structures, Buildings and Facilities; Technical Diagnostics and Strengthening of Steel Structures of Buildings and Facilities; Construction Technology of Buildings and Structures; Design of Steel Structures of Buildings and Structures of Significant consequences class.*

4. Situation analysis

In traditional learning, students often perform calculations, drafting, and modeling (including creating a 3D model of a structure on a computer). However, modeling, analytical models, and drawings are treated as separate stages, which complicates the development of integrated skills (see Fig. 1).

As a result, students have to learn calculation, drafting, and modeling separately - and only later 'assemble' these elements together in practical applications.

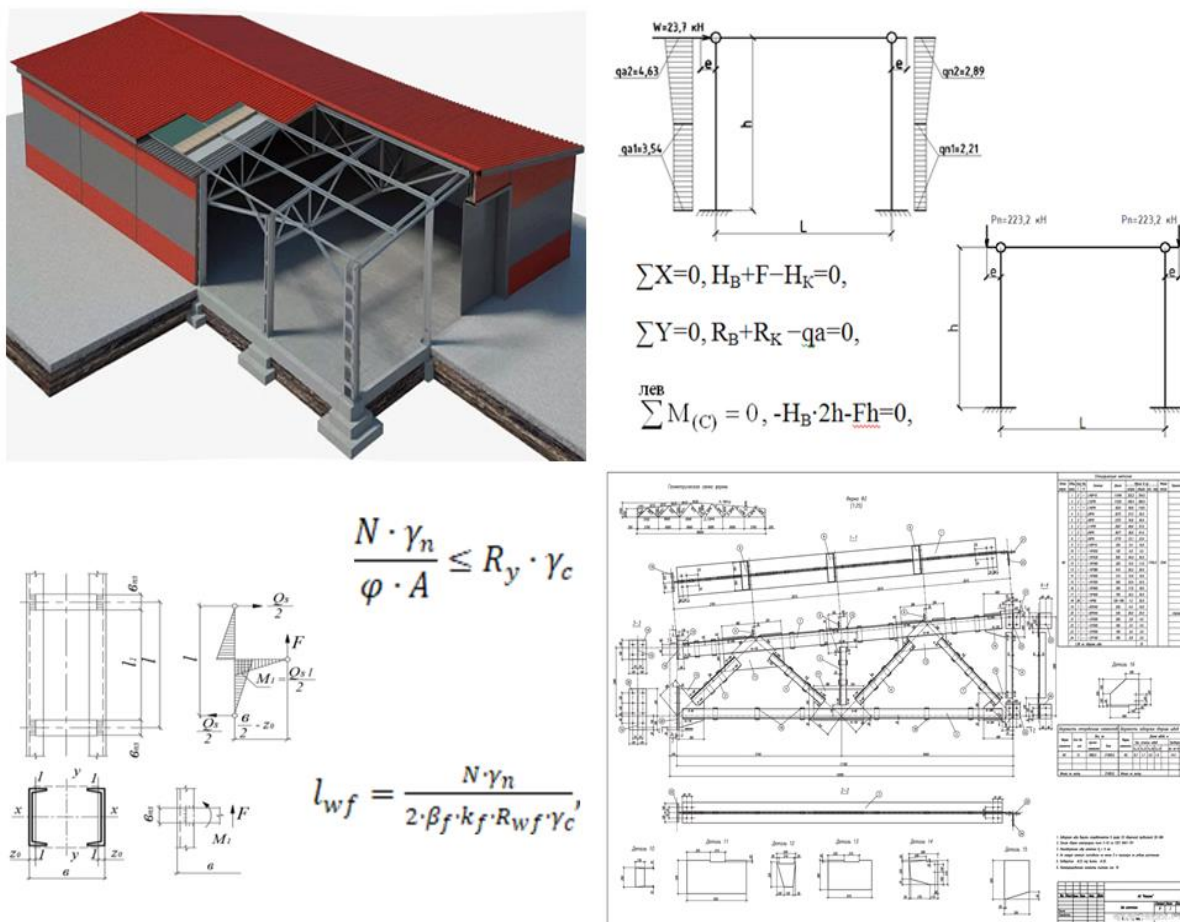


Fig. 4.1. «Traditional» approach: separate stages of calculation (analysis) and design

In real-world design practice, these stages are integrated within a BIM environment. There is an important question: how can the integration of calculation and modeling be ensured within the educational process?

4.1. Main configurations of Tekla Structures

Let us begin with an overview of the main configurations of Tekla Structures (Trimble) - these are different packages selected by an organization depending on its specific tasks [1–5]. Tekla Structures always operates with a single (BIM) model, but each configuration provides access only to a specific set of tools (see Fig. 4.2). Tekla Structures Drafter configuration (see Fig. 4.2a) is used

as an auxiliary module: it offers limited modeling capabilities, focusing on drawings and documentation preparation. Tekla Structures Steel Detailing (see Fig. 4.2b) is intended for steel structure designers: creates KM, KMD models, provides a comprehensive library of steel profiles, joints, and connections, and supports automatic generation of drawings (KM, KMD) and material specifications. Tekla Structures Concrete Detailing (see Fig. 4.2c) is designed for cast-in-place and precast reinforced concrete structures: creates KZ, KZV models, reinforcement of elements (manual and automatic) and provides drawings of reinforced concrete structures and reinforcement specifications. Tekla Structures Engineering (see Fig. 4.2d) is intended for structural engineers: creates analytical models, exports to structural analysis software (SCAD, LIRA, Robot, SAP2000), and verifies load-bearing capacity of elements directly in *Tekla*. Tekla Structures Full (All Configurations) version includes all available features (steel, reinforced concrete, analytical modeling, and project management tools. See Fig. 4.2e).

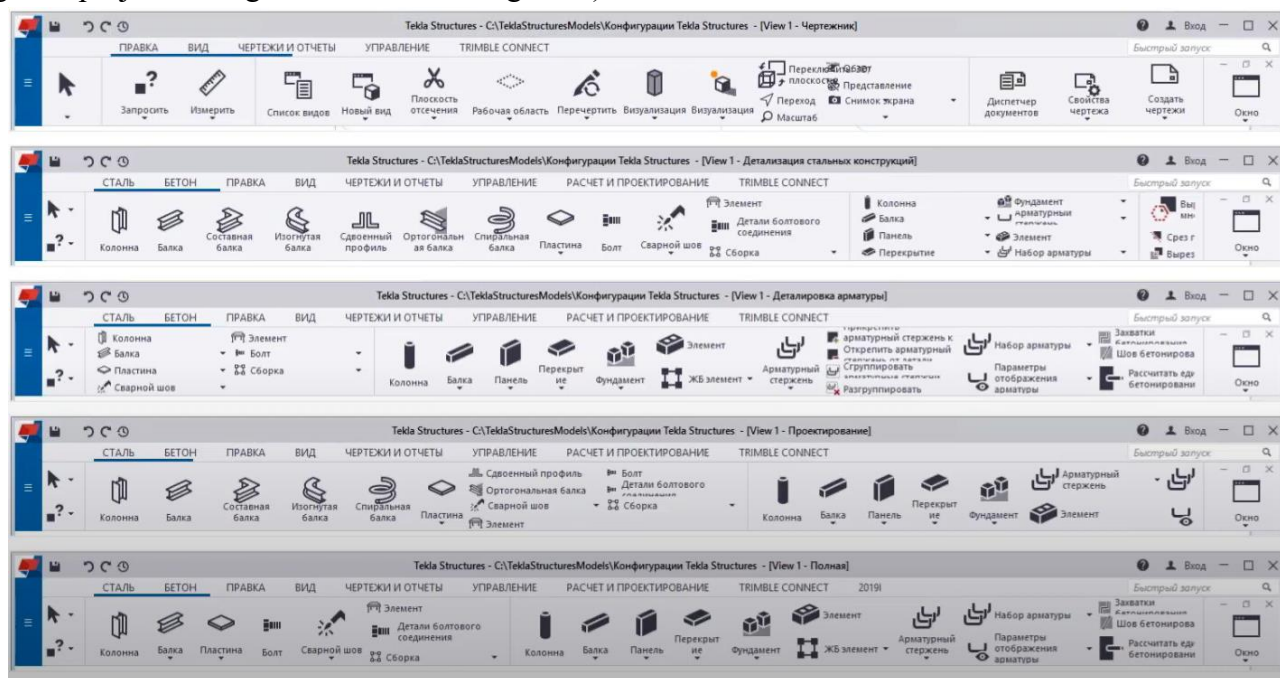


Fig. 4.2. Main toolbars for configurations

Tekla Structures (Trimble) [1]

There are also other configurations used, such as: Tekla Structures Precast Concrete Detailing - specialized for precast reinforced concrete; Tekla Structures Rebar Detailing - a narrowly focused version for creating reinforcement drawings; Tekla Structures Project Management - intended for project managers; Tekla Structures Contractor - designed for contractors and construction companies; Tekla Structures Viewer (Free) - for viewing models in *.tekla* format only [1–3].

4.2. Structural analysis and design of steel structures in Tekla Structures

After analyzing a technical assignment in Tekla Structures, a conceptual model is created, which serves as the basis for detailing and subsequent generation of structural drawings (KM) [2]. This stage involves making initial design decisions, which includes:

- selection of materials - steel grades according to relevant standards (DSTU, EN);
- adoption of technical solutions - types of joints (bolted, welded, combined), methods of assembly and connection;
- definition of profiles - I-beams, channels, angles, pipes, plates, and their standard series;
- specification of geometry - element dimensions, spacing between columns and beams, story heights;

- placement of openings - holes, service recesses, and penetrations for engineering systems;
- other model parameters - erection tolerances, initial reference points, coordinate system, and grid axes.

The second stage of designing the structural steel (KM) section in *Tekla Structures* involves framing (creating the main elements - columns, beams, girders, trusses, and bracings, see Fig. 4.3) according to the grid layout [5]. At this stage, preliminary cross-sections are assigned to each element, which may later be refined. Preliminary joints are defined by specifying connection types (e.g., bolted flange joints, splice plates, welded seams). The model is organized by levels and grid axes, ensuring correct construction within the coordinate system and proper alignment with the reference grid.

Next, the analytical (calculation) model is created in *Tekla Structures* [3]:

- Open the ‘Analysis & Design’ tab - Analysis Models and Design (see Fig. 4.4);
- Click ‘Create’, then select the SCAD_Tekla application to export the analytical model to SCAD Office (see Fig. 4.5).

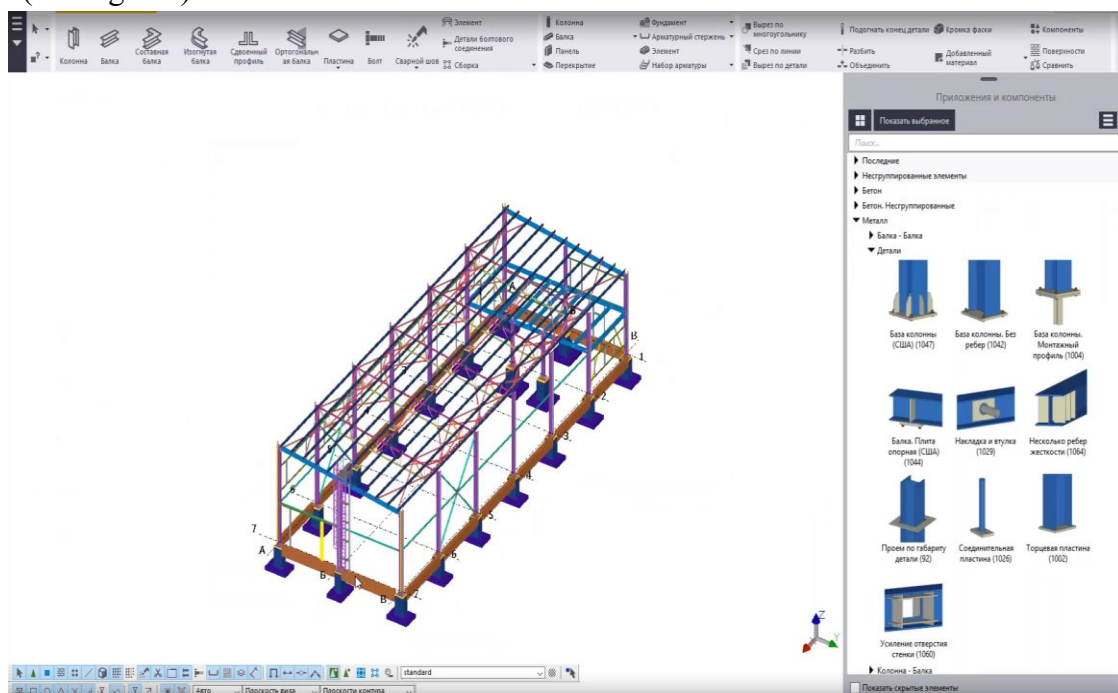


Fig. 4.3. Analytical model in Tekla Structures

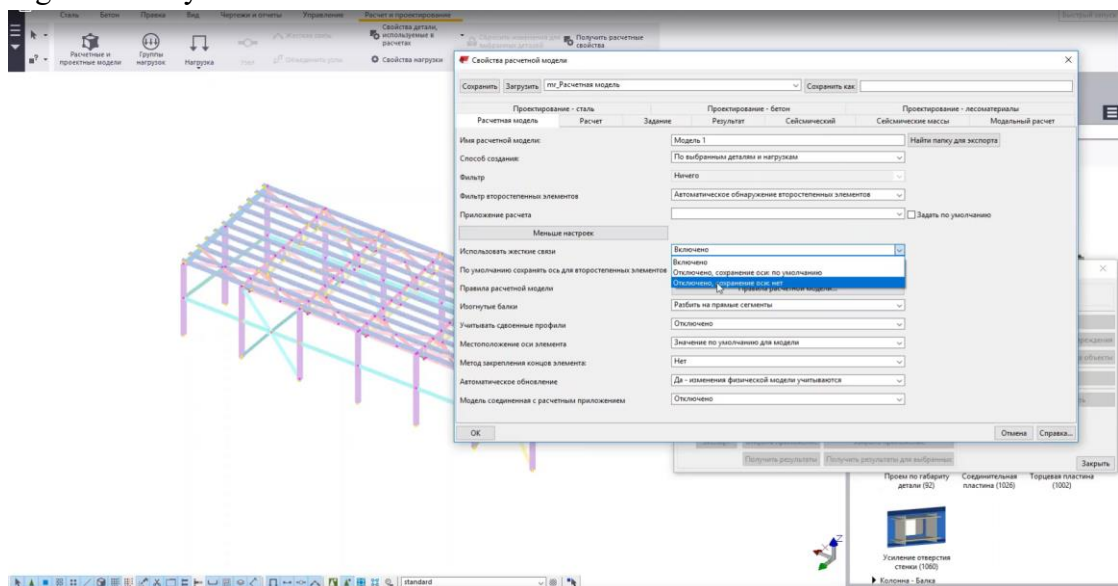


Fig. 4.4. Analysis & Design tab - Analysis Models and Design

Next, the export settings to SCAD Office are configured. Click 'Export' to open this window. In this window, specify the types of elements, joints, supports, loads, and cross-sections from *Tekla Structures*.

When working with structural analysis applications (including SCAD Office), the export process can be organized in different ways - either by exporting the entire model at once, or by fragments when analyzing local joints, individual beams, cantilevers, roof segments, or separate frames.

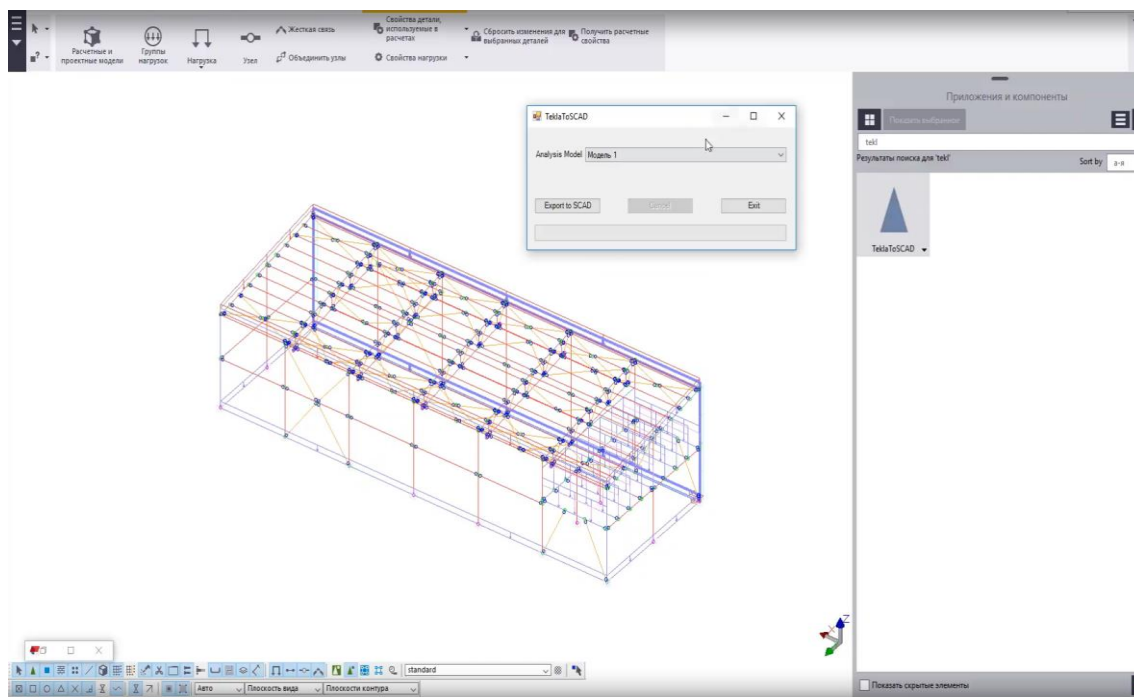


Fig. 4.5. Creating an analytical model, configuration of export settings, and export to SCAD Office

Open the analytical model in SCAD Office (see Fig. 4.6) [21] and, if necessary, make adjustments such as support conditions, stiffness parameters, and others. Export the model from Tekla Structures or create loads, load combinations, and load cases directly in SCAD Office.

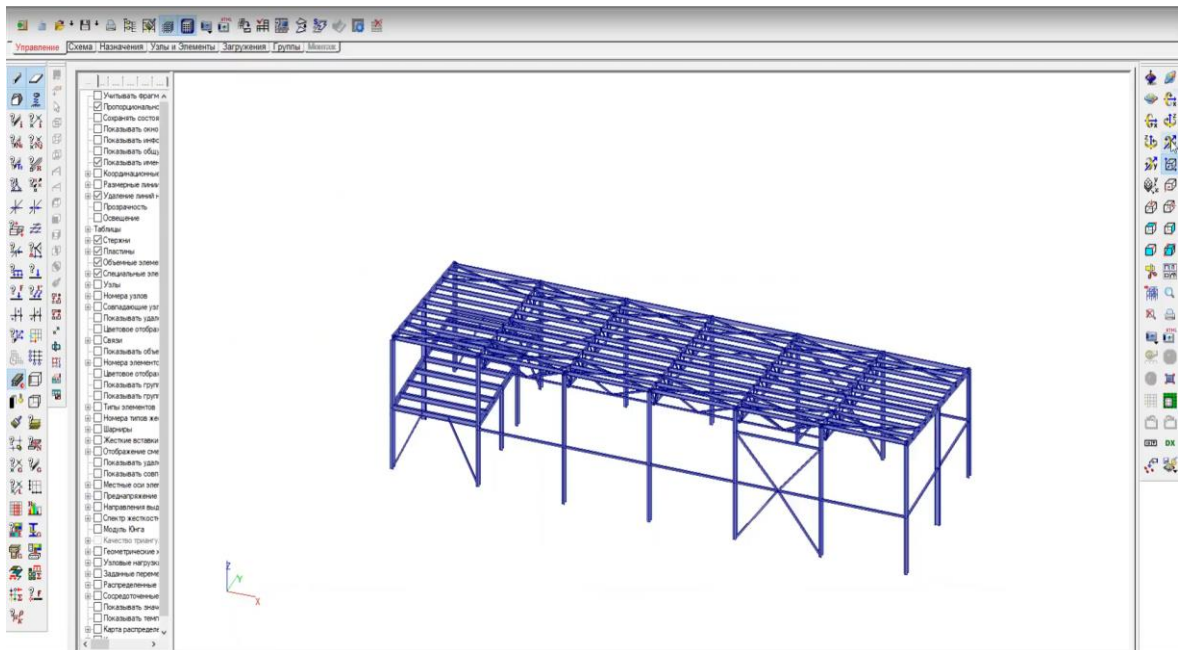


Fig. 4.6. Analytical model in SCAD Office

In *Tekla Structures*, the export of an analytical or calculation model to LIRA-SAPR [20] (as well as to SCAD Office) is performed using special plugins/converters that ensure the accurate transfer of data (see Fig. 4.7).

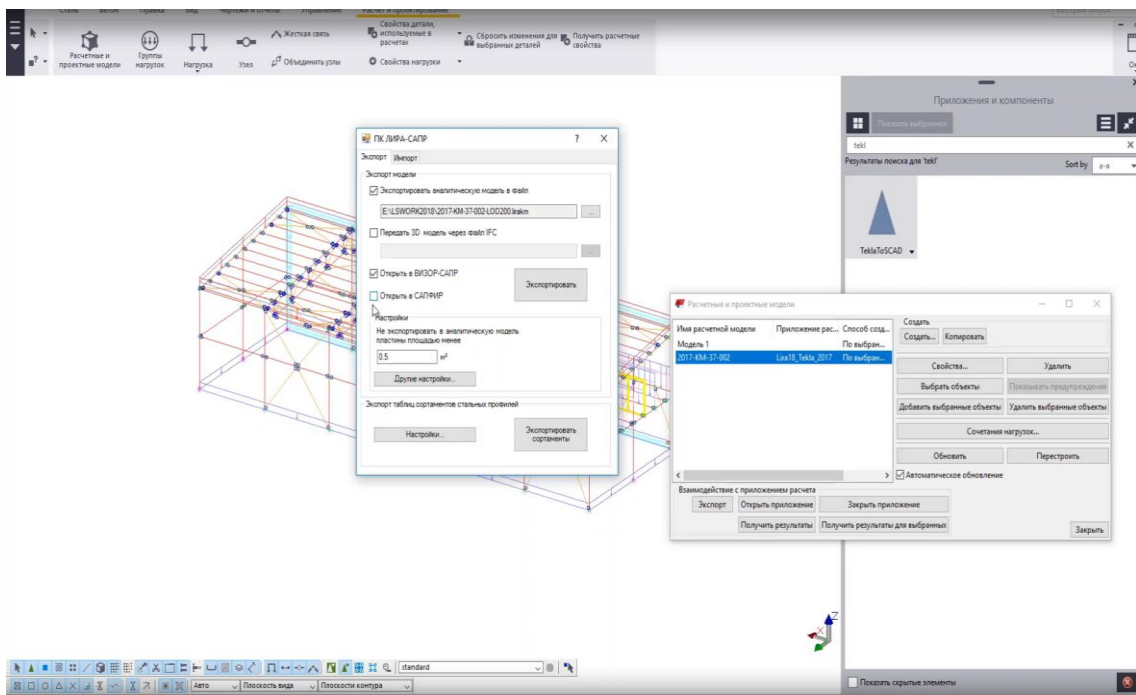


Fig. 4.7. Creating an analytical model, configuration of export settings, and export to LIRA-SAPR (VIZOR-SAPR for steel structures)

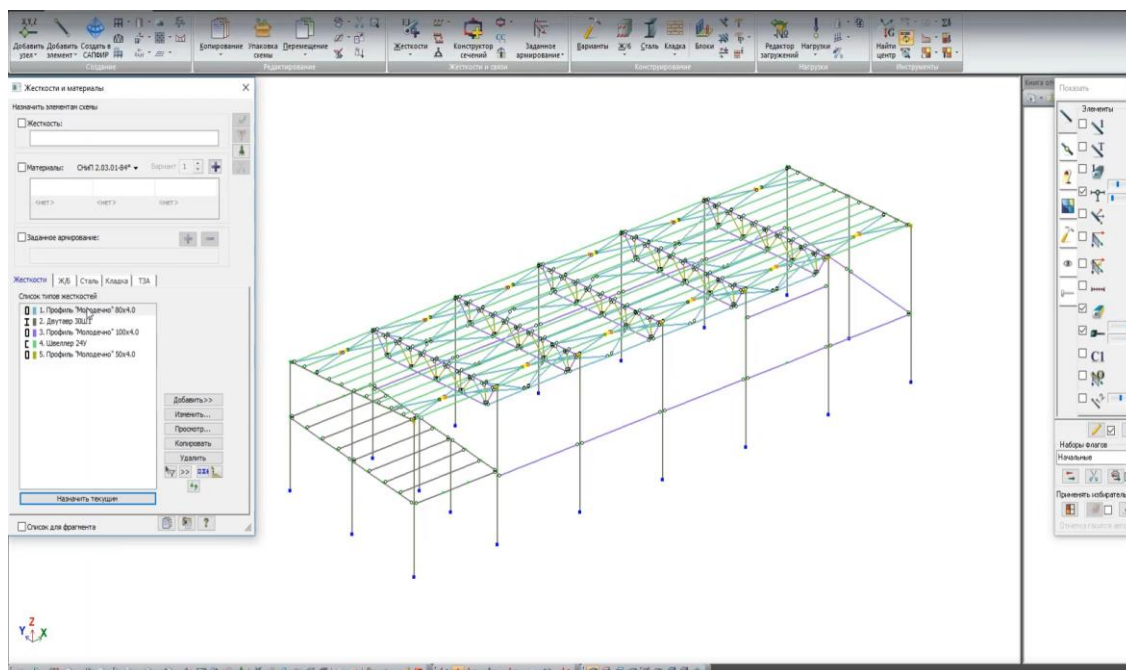


Fig. 4.8. Analytical model in LIRA-SAPR

In *Tekla Structures*, open the Analysis - Analysis & Design Models tab. Create a new model for export, select LIRA as the target application, and configure the export options accordingly.

In *Tekla Structures*, there is a built-in analytical grid that serves as the foundation for static analysis. On the Analysis - Load Application tab, you can add different types of loads: permanent, variable, and special [2]. Then, load groups are created and combined into analytical combinations. *Tekla* automatically generates these analytical load combinations, which are then transferred to the structural analysis phase. Select the Analysis - Run Analysis / Solve tab. *Tekla Structures* uses its own calculation engine for analyzing the analytical model (the finite element method (FEM) for frame and plate structures). After the analysis is completed, the following results can be obtained: internal forces in elements (N , Q , M - axial, shear, and bending); deflections of elements and nodes; stresses and section utilization factors. For accurate calculations according to national standards, specially developed plugins are required.

4.3. Creating connections in Tekla Structures

Tekla Structures includes a built-in library of standard joints, which comprises corner and T-shaped connections, beam-to-column joints, and connections with fitting elements (plates, angles, brackets, welds) [5].

Advantages of using standard joints: rapid modeling of structures; reduced risk of errors in the placement of bolts and fittings; compliance with international standards (EN, ISO).

After creating the connection, it is possible to preview the model in the 3D window of *Tekla Structures* (see Fig. 4.9):

- display the beam grid with all connection nodes;
- check the correct placement of fittings, bolts, and welds;
- perform a clash and clearance check between model elements.

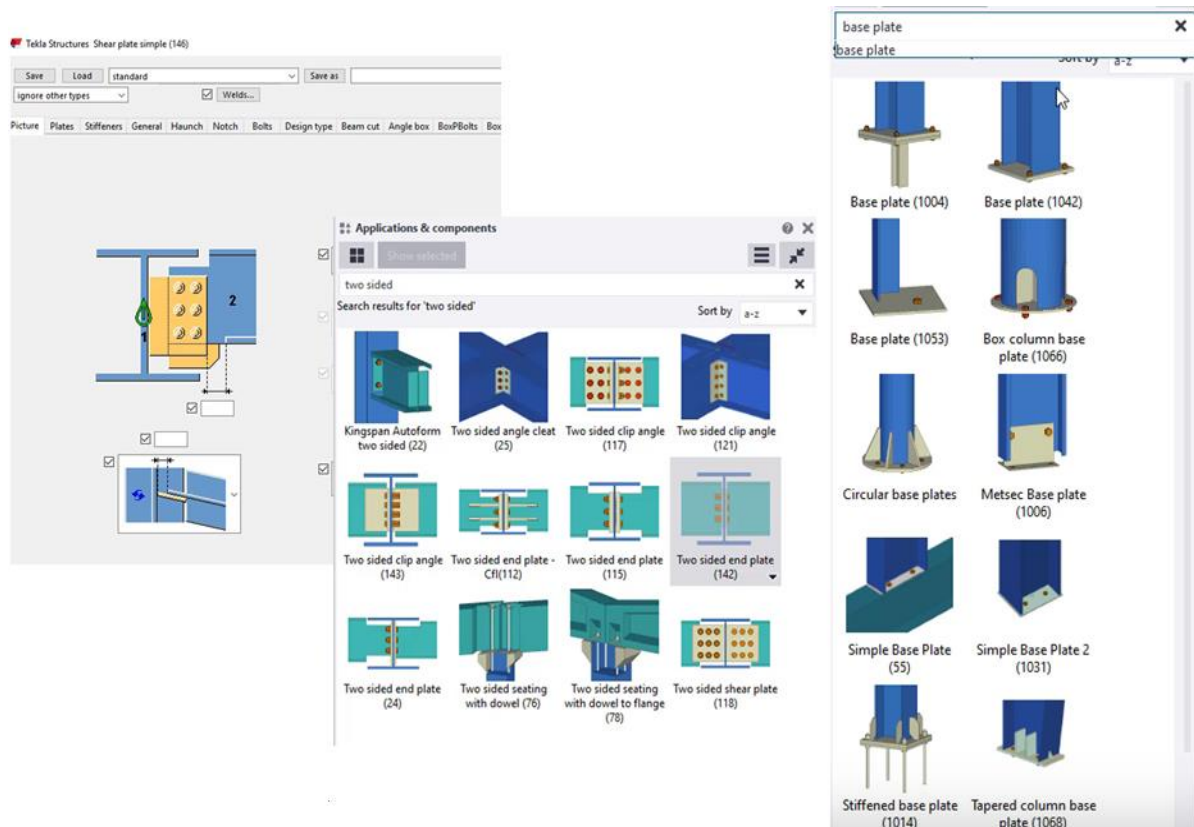


Fig. 4.9. Modeling connections using system components in Tekla Structures

Fittings (plates, brackets) are added to the joints to ensure strength and stiffness of the connection.

Configuration process:

- select the joint in the model;
- open *Node properties*;
- add a fitting plate from the library or create a custom one;
- specify thickness, material, and dimensions of the plate;
- set the placement of fittings relative to beams and columns.

Bolts are used to connect structural elements without welding.

Bolt configuration in joints:

- select the bolt type (e.g., high-strength, class 8.8, 10.9, etc.);
- specify the bolt diameter and length;
- choose the bolt arrangement pattern (rows, groups, symmetric layout);
- set the spacing between bolts and the edge distances of the elements;
- *Tekla* will automatically display the bolts in the 3D model and on the drawings.

Welds are used for permanent (non-detachable) connections.

Configuration process:

- select the weld type (fillet, butt, or T-joint);
- specify the weld length and location;
- enter the welding parameters (e.g., electrode class, weld thickness);
- *Tekla* will automatically display the welds in the 3D model and on the drawings.

4.4. Creating drawings in Tekla Structures

Tekla Structures enables create automatically working drawings for steel structures (KM) and fabrication details (KMD) based on a 3D model. In the model, select the desired structure or an individual element (e.g., a column, beam, or truss). In the menu *Create Drawing* → *Assembly / Part*, the corresponding erection drawing is automatically generated. Assembly Drawing is a drawing of a connection or assembly (for example, a column with attached beams). Single Part Drawing is a drawing of an individual component for manufacturing (plates, bolts, beams) [5].

The drawing retrieves information on materials, dimensions, geometry, and element positioning directly from the 3D model.

In *Tekla Structures*, front, plan, isometric, and detailed views of the structure can be created automatically or manually. The views can be scaled for better readability, Primary and auxiliary views (such as *Section View* and *Elevation*) can be added.

Annotations in *Tekla Structures* automatically display:

- element numbers;
- lengths, widths, and thicknesses;
- materials and bolted connections;
- element positions and markings according to specifications.

Tekla also generates specifications (Parts List / BOM) - detailed material tables that include:

- number of parts;
- length and profile type;
- material and steel grade;
- llement weight.

These specifications are automatically linked to the drawing (see Fig. 4.10).

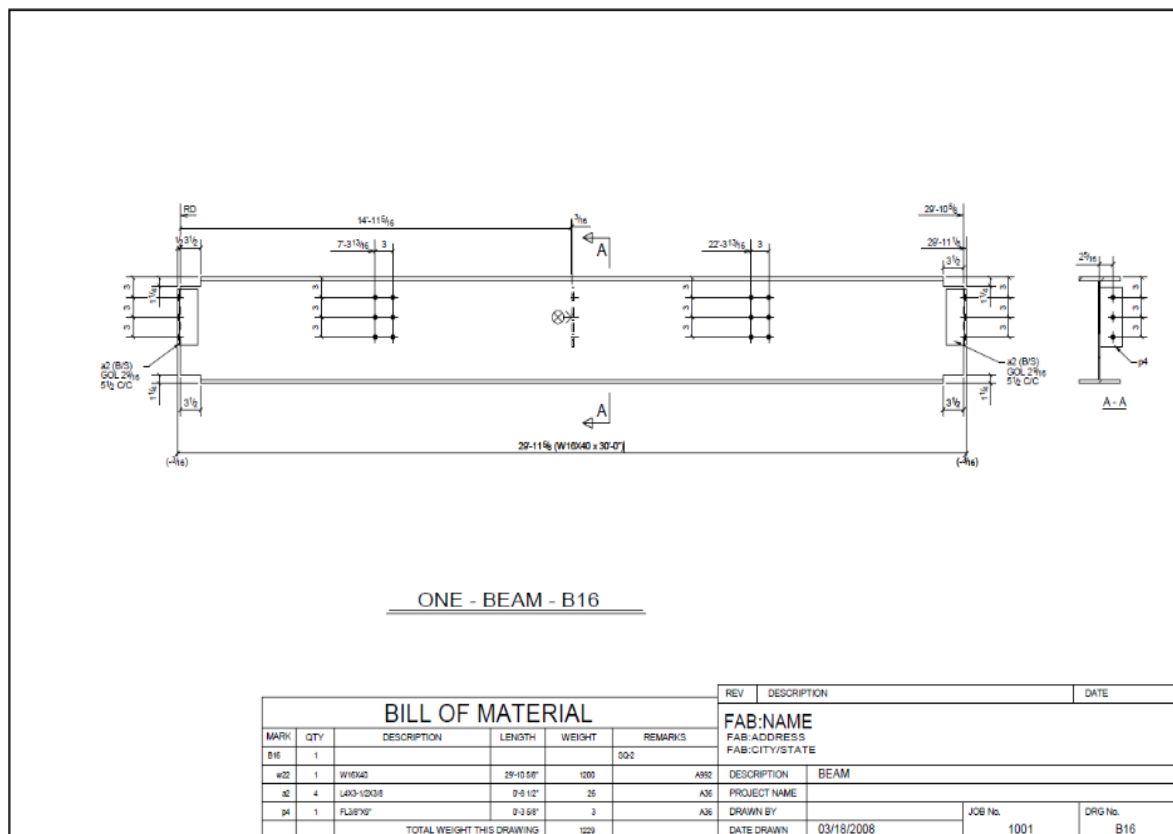


Fig. 4.10. Example of a beam drawing automatically generated by Tekla Structures.

- Advantages of automatic drawing generation in Tekla Structures:
- the number of errors in drawings is reduced due to synchronization with the 3D model;
 - the process of design and documentation preparation is faster;
 - drawings are easily updated when the model changes - all annotations, views, and specifications are automatically updated.

4.5. 'Ukraine' Environment for Tekla Structures

To generate Project documentation in compliance with national standards, the 'Ukraine' environment must be applied [4]:

- a set of catalogues, templates, and settings for Tekla Structures;
- operates in accordance with Ukrainian standards and regulations;
- suitable for the Project sections: KM, KMD, KB, KBV.

For steel structures, the material library includes steels defined in accordance with national DSTU and GOST standards (see Figure 4.11).

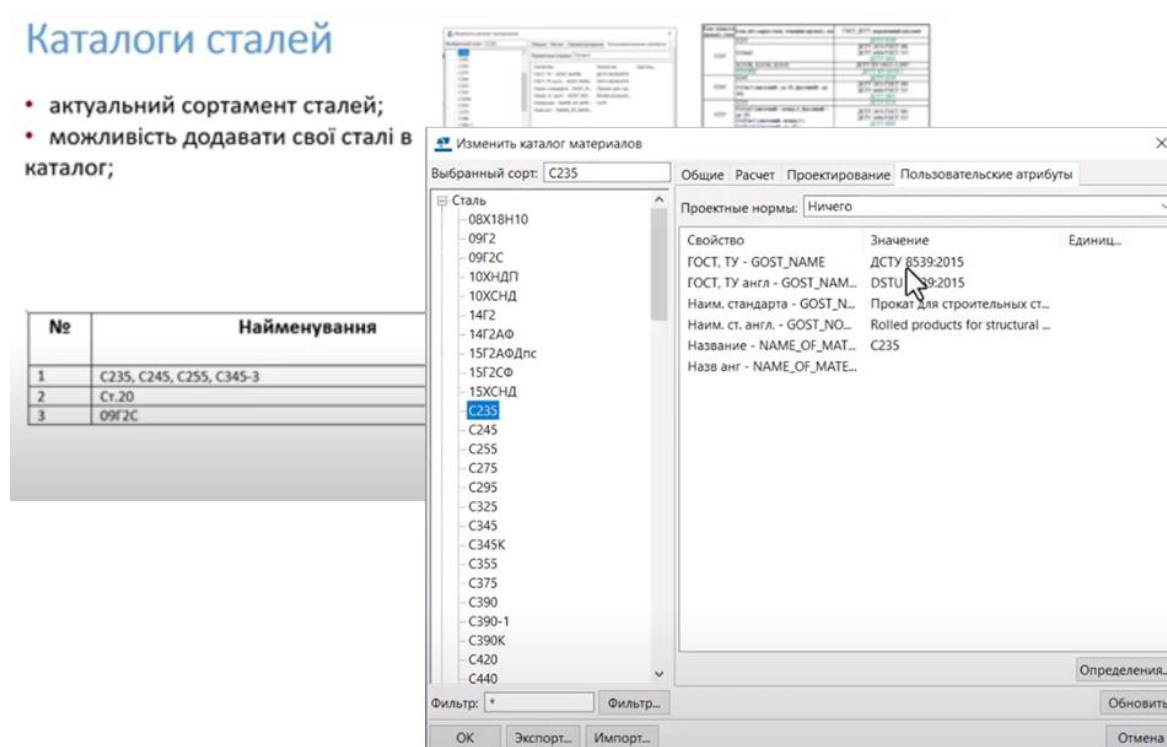


Fig. 4.11. Catalogues and materials, 'Ukraine' environment

Profile catalogues in the 'Ukraine' Environment (see Fig. 4.12)

- I-beams;
- channels;
- angles (equal and unequal legs);
- pipes (round, square, rectangular);
- rails for cranes and railways, etc.

There are also catalogues for bolts, nuts, and washers (see Fig. 4.13).

The environment provides automatic generation of specifications displayed on drawings - 'Structure Group' column used for classification (see Fig. 4.14). In addition, the catalogues for reinforced concrete structures are available, allowing users to add custom concrete grades, as well as reinforcement and embedded part specifications.

Каталоги профілів

- актуальний сортамент профілів;
- можливість додавати свої профілі в каталог;

№	Найменування	ДСТУ
1	Двутаври сталеві гарячекатані	ДСТУ 8768:2018
2	Швеллери сталеві гарячекатані	ДСТУ 3436-96
3	Кутяки сталеві гарячекатані рівнополочні	ДСТУ 2251:2018
4	Кутяки сталеві гарячекатані нерівнополочні	ДСТУ 8769:2018
5	Двутаври гарячекатані для кріплення цвяхів	ДСТУ 8807:2018
6	Труби сталеві електроізольовані	ГОСТ 30754-93
7	Труби сталеві бітумні гарячекатані	ГОСТ 9732-78
8	Профіль круглий	ДСТУ 4736:2007
9	Профіль квадратний	ДСТУ 4746:2007
10	Швеллери сталеві шпунтовані	ДСТУ 8806:2018
11	Кутяки сталеві шпунтовані	ДСТУ 2254-93
12	Профіль сталевий шпунтований	ДСТУ 8.2.9-8-95
13	Профіль сталевий гарячекатаний холоднокатаний	ДСТУ 4747:2007
14	Рельси кранові	ДСТУ 2484-94
15	Рельси для залізничних вагонів ширини колії стандартної системи в напрямку транспортів	ДСТУ 4349:2004
16	Рельси металеві дорожні типу Р60 для вуличного промислового транспорту, трамваїв та метрополітену	ДСТУ 2539-94
17	Труби сталеві профільні	ДСТУ 8840:2018
18	Вироби з сталевих труб і сталевих заклепок декоративного призначення для будівництва великих промислових об'єктів	ДСТУ 8802:2018

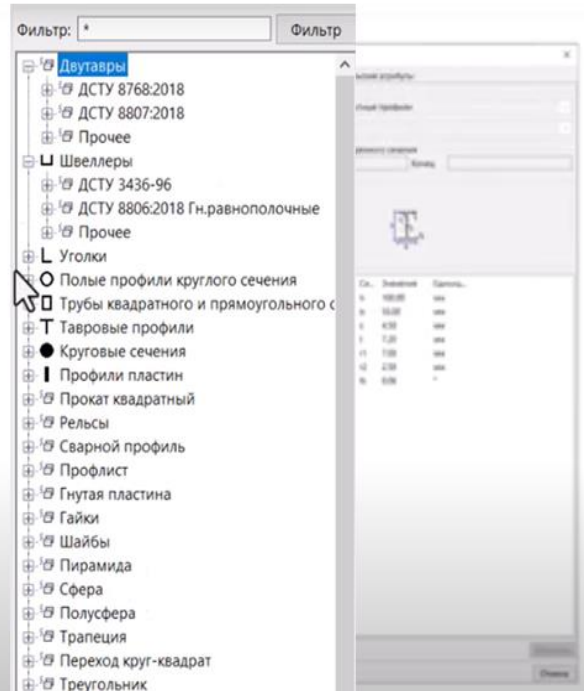
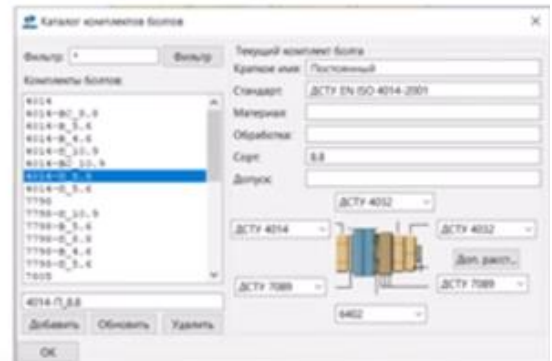


Fig. 4.12. Profile catalogues, 'Ukraine' environment

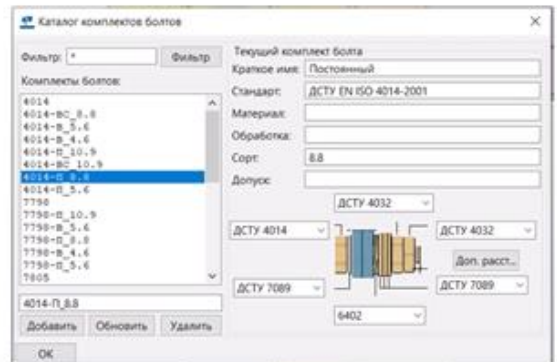
Каталоги болтів

- актуальний сортамент;
- можливість додавати свої болти, гайки, шайби і каталогів болтів;
- кольорове відображення болтів по типу високоміцні заводські, високоміцні монтажні, прості заводські, прості монтажні
- відображення символів болтів на кресленнях



Каталоги болтів

- актуальний сортамент;
- можливість додавати свої болти, гайки, шайби і каталогів болтів;
- кольорове відображення болтів по типу високоміцні заводські, високоміцні монтажні, прості заводські, прості монтажні
- відображення символів болтів на кресленнях



№	Комплект болтів	Болти	Гайки	Шайби	Клас міцності
1	Тимчасові	ДСТУ EN ISO 4014:2001	ДСТУ ISO 4032:2002	ДСТУ EN ISO 7089:2022	4.6
					5.6
2	Постійні				5.6
					8.8
3	Високоміцні	ГОСТ 22356-77	ГОСТ 22356-77	ГОСТ 22356-77	8.8
					10.9

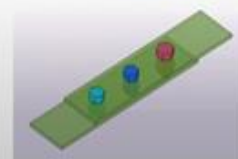
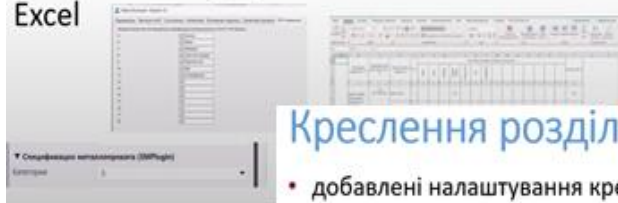


Fig. 4.13. Bolt catalogues, 'Ukraine' environment

Відомості і специфікації по розділу КМ.

Специфікація металопрокату

- СМ – створюється за допомогою плагіна «SMPlugin2UA»,
- дані для специфікації металопрокату заповнюються в налаштуваннях проекту і властивостях головної деталі збірки,
- СМ вноситься на креслення одним кліком і експортуються автоматично в Excel



Специфікація металопрокату

Назва профілю	Клас	Назва профілю	№	Розміри металу в таблиці елементів конструкції, мм										Значення			
				h	b	t	r	s	l	l ₁	l ₂	l ₃	l ₄				
Листовий профіль	С205	Листовий профіль	1														
Листовий профіль	С205	Листовий профіль	2														
Листовий профіль	С205	Листовий профіль	3														
Листовий профіль	С205	Листовий профіль	4														
Листовий профіль	С205	Листовий профіль	5														
Листовий профіль	С205	Листовий профіль	6														
Листовий профіль	С205	Листовий профіль	7														
Листовий профіль	С205	Листовий профіль	8														
Листовий профіль	С205	Листовий профіль	9														
Листовий профіль	С205	Листовий профіль	10														
Листовий профіль	С205	Листовий профіль	11														
Листовий профіль	С205	Листовий профіль	12														
Листовий профіль	С205	Листовий профіль	13														
Листовий профіль	С205	Листовий профіль	14														
Листовий профіль	С205	Листовий профіль	15														
Листовий профіль	С205	Листовий профіль	16														
Листовий профіль	С205	Листовий профіль	17														
Листовий профіль	С205	Листовий профіль	18														
Листовий профіль	С205	Листовий профіль	19														
Листовий профіль	С205	Листовий профіль	20														

Креслення розділів КМ і КМД

- додані налаштування креслень для розділів КМ, КМД
- основні надписи на креслення на Українській мові



```

1_KMD_Vidomist_metaloprokatu_Metizy_motazhni_na_proekt
1_KMD_Vidomist_metaloprokatu_Metizy_na_proekt
1_KMD_Vidomist_vidpravnykh_elementiv_desyatychni_chysla
1_KM_Navantazheniya_na_fundamenti
1_KM_Vidomist_elementiv_marka
1_KM_Vidomist_elementiv_prefix
1_KM_Vidomist_elementiv_skladnyy_pereziz_marka
1_KM_Vidomist_elementiv_skladnyy_pereziz_prefix
1_Tyutynyy_lyst
1_Zahalni_dani
1__Общий_фнд__
1_KMD_Vedomost_otp_elementov-2_kolonni-Целье_числа
1_KMD_Vedomost_otp_elementov-3_kolonni-Целье_числа
1_KMD_Vedomost_otp_elementov-Целье_числа
    
```

Fig. 4.14. Specifications, 'Ukraine' environment

5. Discussion questions

1. What are the main advantages of BIM platforms compared to traditional 2D design?
2. How does Tekla Structures integrate into the overall building design cycle?
3. Why is it important to combine 3D modeling, analysis, and drawing generation in a single environment?
4. How should the transition from a BIM model to a structural analysis model be performed correctly? What common mistakes may occur?
5. Which software packages for steel structure design have you already used?
6. How does traditional design (AutoCAD, manual methods) differ from the BIM approach?
7. What are your expectations from working with Tekla Structures compared to other CAD systems?
8. Why is it critically important to consider joint stiffness in structural analysis?
9. What are the differences between an automatically generated analysis model and a manually simplified model?
10. In which cases is it reasonable to perform parallel verification of Tekla results in SCAD Office or LIRA-SAPR?
11. How is compliance of Tekla models with Ukrainian (DBN, DSTU) and European (Eurocode) standards ensured?
12. Can local databases of profiles and materials be adapted to meet current standards?

13. What are the advantages of comparing calculation results according to DBN and Eurocodes?
14. What steps are required to build a basic 3D model of a steel frame in Tekla?
15. How does Tekla help automate routine operations (marking, specifications, drawing templates)?
16. Which Tekla tools are the most convenient for creating working documentation?
17. How can the adequacy of the analytical model be evaluated?
18. What is the benefit of performing a sensitivity analysis of the model to parameter changes?
19. What examples of research studies could be based on digital Tekla models?
20. How can teamwork on a single BIM model be organized?
21. Which Tekla tools help avoid clashes or conflicts between different parts of the model?
22. How should a 'change log' be properly maintained in project documentation?
23. How can Tekla export to IFC or SCAD/LIRA formats enable interoperability between software?
24. Is it enough for a student to know the standard, or is it more important to understand the logic behind its application?
25. How should one prepare for a project defense with well-argued use of standards and codes?
26. Does a student still need manual calculation skills if Tekla already automates most tasks?
27. How can Tekla be integrated with other structural analysis programs (SCAD, LIRA-SAPR)?
28. Can BIM technology fully replace a traditional approach in course and diploma projects?
29. Which skills are more important for a future engineer - the ability to draft manually, or to work with a digital model?

6. Possible solution strategies

Integration of Tekla Structures into educational programs:

- special courses and practical training in BIM technologies within student curricula;
- use of Tekla Structures as a core tool for the development of course and diploma projects.

Development of methodological materials

- creation of step-by-step instructions for students;
- preparation of video tutorials and training examples for modeling and analysis;
- development of localized databases of profiles and materials (DBN/EN).

Practical integration of analysis (calculations)

- use of model export to SCAD Office and LIRA-SAPR to verify results;
- comparison of automatic calculations in Tekla with manual computation methods;
- performing sensitivity analysis of parameters (joint stiffness, loads, materials).

Organisation of teamwork

- development of skills for working within a shared BIM model;
- training students in coordination between different specializations (structural engineers, architects, MEP engineers);
- practicing change log maintenance and model version control procedures.

Research

- development of research in the field of design automation;
- testing of new algorithms for the automatic generation of analytical models;
- exploration of methods for integrating DBN and Eurocode standards into digital design environments.

Digitisation of operation

- use of Tekla models for building monitoring during the operational stage;

- training students to work with IFC formats and data exchange between different BIM platforms;
- development of skills in digital asset management and lifecycle support of buildings during operation.

7. Solution (example)

Use of Tekla Structures to design a steel frame of an educational facility (for example, a single-span industrial building or a working platform).

Integration with structural analysis software such as SCAD Office and LIRA-SAPR enables the verification of the analytical model and the accuracy of structural calculations.

Tekla provides a comprehensive BIM approach: from 3D model through structural analysis to the production of working drawings.

Students simultaneously develop modeling, analytical, and graphical competencies, which correspond to modern professional requirements.

The use of structural analysis software ensures that students not only 'click button' but also understand the underlying physics of processes and verify the obtained results.

8. Practical assignment for students

Model a steel frame of a working platform structure (columns, beams, and joints) in Tekla Structures (see Fig. 8.1).

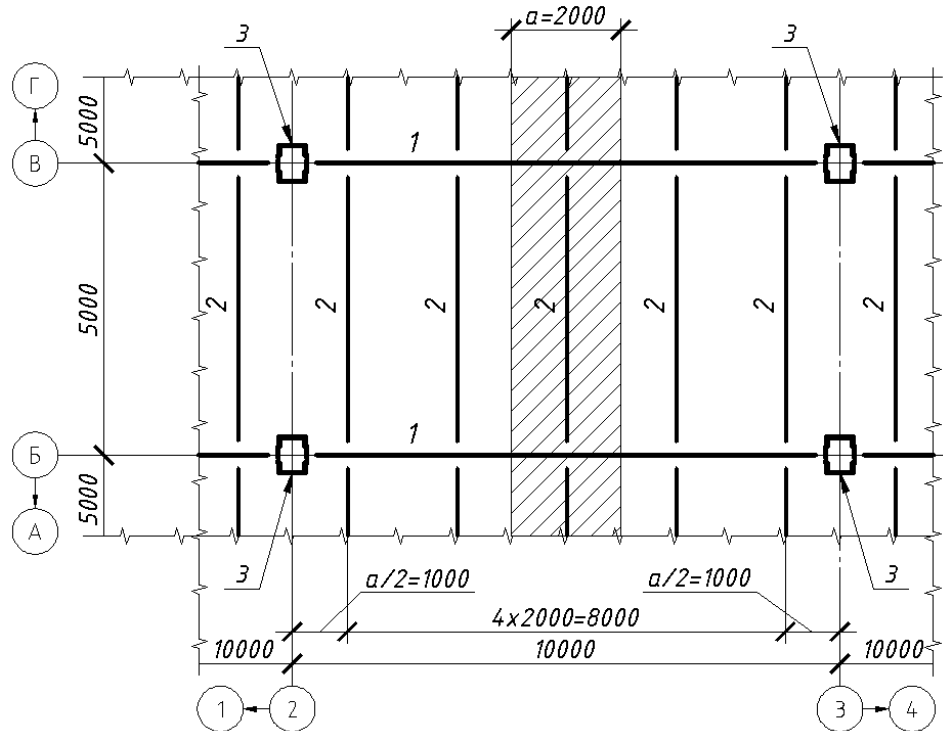


Fig. 8.1. Analytical model of a working platform:
1 – main beam; 2 – decking beam; 3 – column.

Assign materials in accordance with [6–8, 14, 15]. Create an analytical model in Tekla Structures and export it to SCAD Office.

Perform load collection in accordance with [11]. Carry out a static analysis to obtain internal forces and deflections. Select cross-sections according to the analysis results, with regard to [6]:

- decking beams;
- main beam;
- centrally compressed lattice column.

Perform connection and joint design calculations according to [6].

Generate fabrication drawings of the following elements (decking beams; main beam; centrally compressed lattice column).

The assignment as a case study is ‘Design of the steel frame for a working platform using Tekla Structures and SCAD Office’.

It is required to design a working platform shown in Fig. 8.1. The frame layout is based on the following initial design data:

- column spacing in the transverse direction – ___ m;
- column spacing in the longitudinal direction – ___ m;
- elevation of the platform deck top – ___ m;
- design live load on the platform – ___ kPa;
- type of decking – reinforced concrete slab (monolithic) or steel deck (chequered plate).

Guidelines for individual assignment

Build a 3D model of the frame in Tekla Structures. Perform a static analysis in SCAD Office. Justify the selection of column and beam cross-sections. Prepare fabrication drawings of the elements (decking beams, main beam, and centrally compressed lattice column) and a material specification.

The student performs the entire design cycle independently, but the scope of the task is reduced (for example, modeling a single frame instead of the entire structure). Expected results: 3D model, structural analysis, and drawings of one selected element.

Guidelines for teamwork. Roles distribution:

modeling specialist – creates the model in Tekla Structures; calculation specialist – develops the analytical model and performs the structural analysis in SCAD Office; designer – prepares drawings and material specifications; analyst – prepares the report and compares the results obtained from Tekla Structures, SCAD Office, and manual calculations.

The final outcome is a single comprehensive report and a presentation of the results.

9. Discussion and reflection

Tekla Structures as a BIM platform enables students not only to master modeling and analysis tools, but also to critically reflect on modern design approaches. It is important to discuss how the role of an engineer evolves in the digital environment, which competencies become essential, and what challenges arise when integrating national and international standards into engineering practice.

Questions to discuss:

1. How does traditional design differ from the BIM-oriented approach?
2. What challenges arise when transitioning a BIM model into an analytical model?
3. How can the adequacy of automated analysis be verified in practice?
4. Can the results obtained from software be always accepted without additional verification?
5. What role do standards (DBN, DSTU, Eurocodes) play in working with digital models?
6. The principles of transitioning from a digital model to an analytical model can also be applied to reinforced concrete structures.
7. Experience in verifying automated analysis results is valuable when working with other software packages (SCAD, LIRA-SAPR, Robot).
8. The practice of integrating regulatory requirements into a model can be applied to the design of foundations, bridges, or special structures.
9. Teamwork skills developed in a BIM environment can be easily scaled to interdisciplinary projects (architecture + structures + engineering systems).

10. Conclusions

Does the implementation of a case study using Tekla Structures in the learning process enable students to practically master the key stages of modern steel structure design - from 3D modeling to verification of analytical models in accordance with DBN/DSTU and Eurocode standards?

Has the case study demonstrated the importance of integrating BIM approaches into professional engineering practice and contributed to the development of critical analysis skills in structural analysis?

References

1. Tekla Structures User Assistance. Trimble Solutions Corporation. [Electronic resource]. Available at: [<https://www.tekla.com/ru/solutions/tekla-structures>]. Accessed: 17.09.2025.
2. Tekla Structures. Basic Online Course. [Electronic resource]. Available at: [<https://sapr-art.by/tekla>]. Accessed: 17.09.2025.
3. Systematic Course on Learning Tekla Structures. [Electronic resource]. Available at: [<https://topengineer.ru/education-centr/book/book>]. Accessed: 17.09.2025.
4. Tekla Structures 2023. New features and improvements: Ukraine environment [Електронний ресурс] // Trimble Solutions Corporation. 2023. Available at: https://support.tekla.com/doc/tekla-structures/2023/rel_2023_sp4_new_features_and_improvements. Accessed: 17.09.2025. Detailing of Steel Structures with Tekla Structures. [Electronic resource]. Available at: <https://www.tekla.com/ru/решения/детализация-стальных-конструкций>. Accessed: 17.09.2025.
5. DBN V.2.6-198:2014. Steel structures. Design standards. – Kyiv: Ministry of Regional Development of Ukraine, 2014. – 199 p.
6. DBN V.2.6-98:2009. Concrete and reinforced concrete structures. General provisions. – Kyiv: Ministry of Regional Development of Ukraine, 2009. – 71 p.
7. DSTU B V.2.6-156:2010. Concrete and reinforced concrete structures made of heavy concrete. Design rules. – Kyiv: Ministry of Regional Development of Ukraine, 2010. – 118 p.
8. DBN V.2.1-10:2018. Bases and foundations. General design provisions. – Kyiv: Ministry of Regional Development of Ukraine, 2018. – 36 p.
9. DBN V.2.6-162:2010. Masonry and reinforced masonry structures. General provisions. – Kyiv: Ministry of Regional Development of Ukraine, 2010. – 97 p.
10. DBN V.1.2-2:2006. Loads and actions (with amendments No. 1 of 01.10.2007 and No. 2 of 01.06.2020). – Kyiv: Ministry of Construction of Ukraine, 2006. – 60 p.
11. DSTU B V.1.2-3:2006. Deflections and displacements. – Kyiv: Ministry of Construction of Ukraine, 2006. – 10 p.
12. DSTU B V.2.6-193:2010. Protection of steel structures against corrosion. – Kyiv: Ministry of Regional Development of Ukraine, 2013. – 70 p.
13. DBN V.1.2-14:2018. General principles for ensuring reliability and structural safety of buildings and structures. – Kyiv: Ministry of Regional Development of Ukraine, 2018. – 30 p.
14. DSTU 8855:2019. Determination of the consequence class of buildings and structures. – Kyiv: SE 'UkrNDNC', 2019. – 29 p.
15. DBN V.1.1-12:2014 (with Amendment No. 1 of 01.05.2019). Construction in seismic regions of Ukraine. – Kyiv: Ministry of Regional Development of Ukraine, 2014. – 110 p.
16. EN 1991: Eurocode 1: Actions on structures. - Brussels, 2002.
Recommended for individual learning
17. Eastman C., Teicholz P., Sacks R., Liston K. *BIM Handbook: A Guide to Building Information Modeling*. 3rd ed. - Hoboken: John Wiley & Sons, 2018.

18. Tekla Online Learning Center. [Electronic resource]. Available at: <https://www.tekla.com/learning>. Accessed: 17.09.2025.

19. Karpilovskyi, V.S.; Kryksunov, E.Z.; Mykytenko, M.A.; Perelmuter, A.V.; Perelmuter, M.A.; Fedorovskyi, V.H. SCAD Office: Implementation of SNIIP in Design Software [Electronic resource]. – Kyiv: VPP “Kompas”, 2001. – 215 p. – Available at: <https://lidermsk.ru/media/documents/ed/ed217bddd48b3d1748fb800b21b60538.pdf> Accessed: 17.09.2025.

8. BASICS OF CALCULATION AND DESIGN OF BUILDINGS USING LIRA-FEM SOFTWARE

Levkivskyi D. V.

1. INTRODUCTION

In modern construction design, the role of **BIM (Building Information Modeling)** technologies is rapidly growing, allowing all stages of a building's life cycle to be integrated into a single digital environment. This approach ensures the integrity and consistency of information at all stages, from conceptual design to construction and subsequent operation. **BIM** technologies make it possible to reduce project development times, minimize errors, and improve interaction between project participants.

Thanks to the implementation of digital models of design objects, participants in the construction process can quickly make changes, analyze design options, forecast costs, and plan work schedules. In addition, **BIM** allows you to create 3D visualizations of objects, which contributes to a better understanding of the spatial structure of the future building.

BIM tools not only enable the creation of geometric models, but also include information about materials, technical characteristics, operational properties, and regulatory requirements. One of the most effective tools for implementing **BIM** principles in Ukraine is the **SAPFIR** and **LIRA-SAPR** software packages.

LIRA-SAPR has been renamed **LIRA-FEM** since 2025. Their integration allows you to build complete digital models of buildings with subsequent calculation, design, and generation of project documentation in accordance with current building codes.

Thus, the use of **BIM** technologies provides a new level of design quality, increases the competitiveness of construction companies, and meets the modern challenges of digitalization in the construction industry.

Learning objectives and outcomes:

Course participants will learn how to:

- create an architectural model of a building in the **SAPFIR** environment;
- adapt an existing architectural model of a building based on an **IFC** model;
- create an analytical model of a building, set loads and their combinations;
- select cross-sections of load-bearing building structures;
- check their strength and deformability;
- create a calculation report.

2. OVERVIEW OF THE LIRA-FEM SOFTWARE PACKAGE

LIRA-FEM software implements *Building Information Modeling (BIM)* technology and is focused on the design and calculation of building structures. **BIM** technology is implemented through integration with other architectural, calculation, graphics, and documentation systems (**SAPFIR-3D**, Revit, Tekla, AutoCAD, ArchiCAD, Advance Steel, BoCAD, Allplan, etc.) based on DXF, MDB, STP, SLI, MSH, STL, OBJ, IFC, and other files [1]. This is a powerful engineering system for the calculation, analysis, and design of building structures using the finite element method (**FEM**). The **LIRA-CAD** software was developed by Ukrainian engineers and is widely used in the field of design both in Ukraine and abroad.

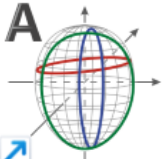

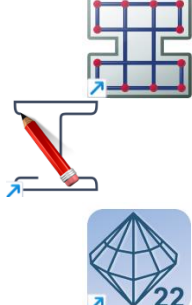


LIRA-CAD is tightly integrated with the **SAPFIR** graphics module, which allows you to create an architectural model of a building, convert it into a calculation diagram, and transfer it to the **LIRA-CAD** for calculations.

SAPFIR (Architectural Design, Form Creation, and Calculation System) is an architectural subsystem for information modeling that is part of the **LIRA-CAD** software package.

The **LIRA-CAD** software package includes additional subprograms that are launched as separate modules from shortcuts and significantly expand the functionality of the system (see Table 2.1).

Together, these modules form a complete system for calculating, designing, and documenting modern buildings and structures.

Table 2.1

	<p>The Avangard module allows you to perform shape generation and topological optimization of structures, which is relevant in tasks of architectural expressiveness or material consumption minimization.</p>
	<p>KM-CAD and Cross-Section Designer enable the preparation of steel structure models with cross-section strength verification, profile selection in accordance with regulatory restrictions, and the ability to create composite cross-sections of any shape.</p>
	<p>LARM-CAD, RS-CAD, and SAPFIR-3D implement tools for automated selection and drawing of reinforcement in reinforced concrete structures, construction of three-dimensional models of objects, and creation of documentation.</p>
	<p>STK-CAD performs modeling of the construction stage, allowing for temporary loads, installation stages, and changes in the spatial rigidity of the structure to be taken into account.</p>
	<p>A module for configuring communication with Autodesk Revit, which integrates LIRA-CAD into the BIM environment by exchanging models via IFC or LiraLink.</p>

3. STAGES OF BUILDING MODELING

SAPFIR software is an integral part of the **BIM**-oriented cycle of building structure design and allows you to create an information model of a building with its subsequent export to the **LIRA-CAD** calculation system. The modeling process is performed in stages and has a clear engineering logic (Fig. 3.1).

Block diagram of building simulation

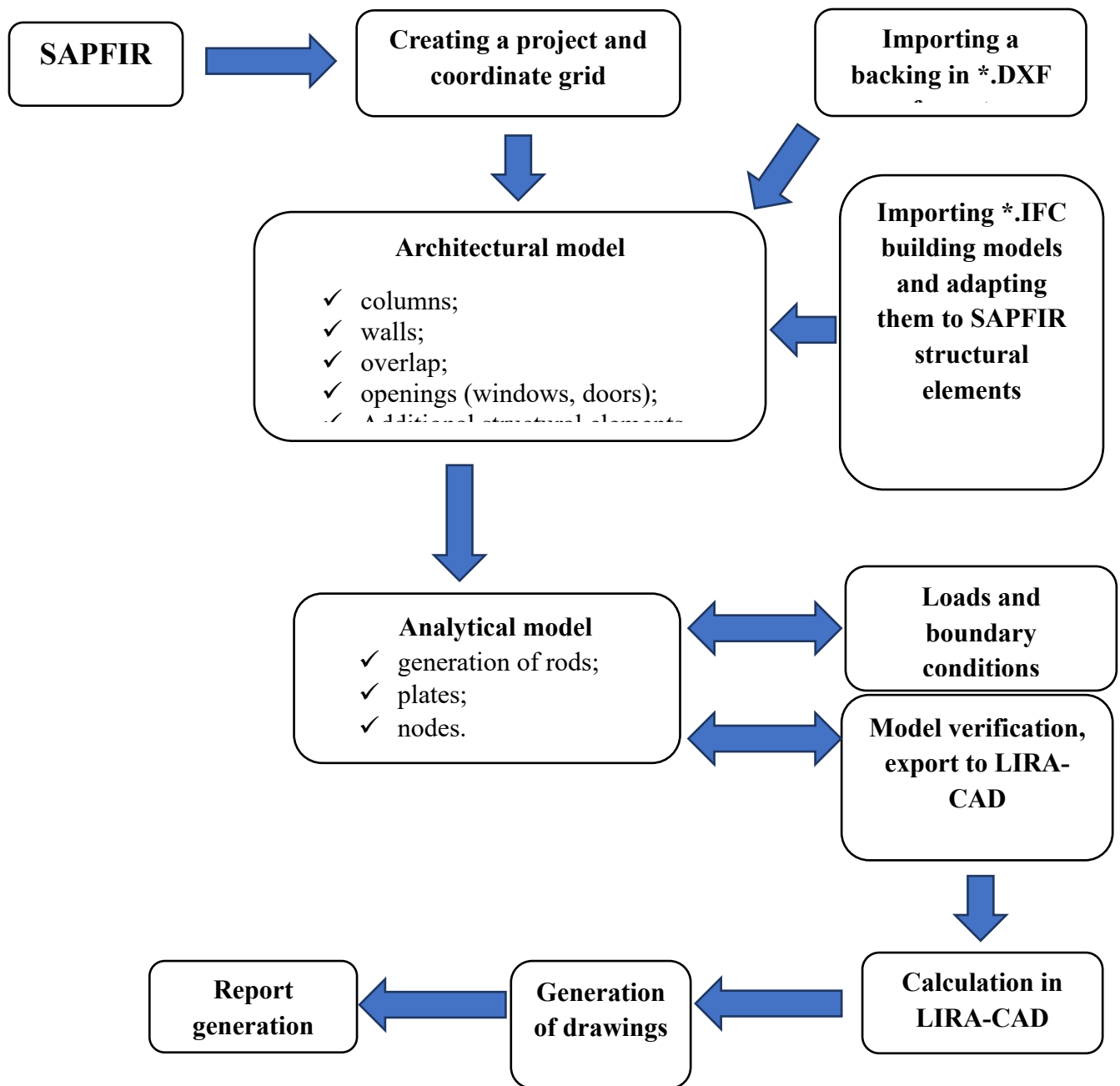


Fig. 3.1 Block diagram of building simulation

3.1 Architectural model

The first stage involves creating a new project and configuring the basic parameters:

- ✓ coordinate grid;
- ✓ floor height;
- ✓ units of measurement;
- ✓ drawing parameters, layers, levels, scales.

If necessary, a “backing” is imported in *.DWG, *.DXF, *.IFC formats, which is used as a basis for spatial modeling (Appendix A).

The modeling process begins with the creation of an architectural model of the building.

- ✓ columns;
- ✓ walls;
- ✓ floors;
- ✓ openings (windows, doors);
- ✓ additional structural elements (ramps, stairs, etc.).

Each element has a set of parameters that determine its physical characteristics and position in space. The model is formed in 2D and 3D space simultaneously. At this stage, topological consistency and logical structuring of the object are ensured.

The **SAPFIR** program allows you to perform the architectural and structural part of the project.

The architectural model contains information about the physical geometry, thicknesses, materials, and mutual arrangement of elements. The analytical model determines the position of axes and planes through which the calculation elements pass. For slabs and walls, **SAPFIR** forms a plane element located in the central plane of the structure (midplane), and for beams and columns, it forms a bar element whose axis passes through the center of gravity of the cross section. These axes and median planes form the framework of the calculation scheme, which is transferred to **LIRA-CAD**. The connection between the architectural and analytical models is important for the accuracy of calculations. All analytical axes must be aligned with each other, and their intersection points must be aligned at common nodes. For a correct calculation scheme, it is necessary to ensure the coaxiality of beams and columns, as well as the exact location of slab planes relative to the axes of elements. It is also necessary to take into account the offsets and ties that depend on the method of supporting the structures. If the analytical axes do not coincide when exporting to **LIRA-CAD**, “broken” nodes are formed, which leads to errors in the calculation.

The **SAPFIR** program provides special tools for controlling the analytical model. In the analytical diagram viewing mode, the user can check the correctness of the axes and nodes, identify inconsistent elements, and automatically correct the connections between slabs, beams, and columns. Before transferring to **LIRA-CAD**, it is advisable to perform a visual check (all building levels are consistent) and use the analytical model collision check function.

After verification, the model is exported to **LIRA-CAD** as a file with the extension *.lir. During export, architectural elements are converted into calculation objects, and the system automatically creates nodes, coordinates, boundary conditions, loads, and other parameters for creating an engineering analysis. It is important to remember that the quality of the links between the architectural and analytical models determines the accuracy of the calculation results and further structural refinement.

3.2 Input data for modeling

The architectural floor plans of the building are provided in Appendix A. Students are also provided with drawings of the building plans in *dxf format. The building has a 3-story frame-monolithic structure, one basement floor, two elevator shafts, and staircases. The load-bearing elements of the building are columns, pylons, stairwell walls, floor beams, and floor slabs. The load-bearing elements are made of heavy concrete reinforced with steel reinforcement. The floor slabs have openings in the locations of stairwells, utility intersections, and ventilation ducts.

The building is subject to influences in accordance with DBN V.1.2-2006:

Permanent loads:

- dead weight of monolithic reinforced concrete structures (determined automatically depending on the geometry and volume weight of materials, taking into account the load reliability factor);

- weight of the floor structure – 50 mm insulation, 60 mm reinforced cement-sand screed, 20 mm ceramic tiles on adhesive – design load 180 kg/m²;

- weight of the covering structures – design load 300 kg/m²;

Long-term loads:

- weight of internal and external partitions, parapets (determined automatically in SAPFIR, taking into account window and door openings, the volumetric weight of materials, and the load reliability factor);

Temporary loads:

- snow load on the roof according to the construction area - calculated value - 160 kg/m²;

- temporary live load on the floor according to - calculated value - 480 kg/m²;

- temporary live load on the roof - calculated value 84 kg/m²;

- wind loads on the building according to the construction area - calculated value - 65 kg/m².

Specific weight of building frame materials, taking into account the load reliability coefficient:

- reinforced concrete – 2500 kg/m³;

- ceramic brick, bulk weight 1800 kg/m³;

- aerated concrete D400, bulk weight 400 kg/m³;

- cement-sand screed 1800 kg/m³.

Task: Build an architectural model of a building in SAPFIR.

The student must develop an architectural and structural model of a three-story building with a basement and flat roof.

The model must be aligned in terms of axes, levels, and structural connections, and must be ready for transfer to the **LIRA-FEM** environment without loss of topology.

4. CREATION OF AN ARCHITECTURAL MODEL OF A BUILDING IN SAPFIR

For convenience when working with the **SAPFIR** program, the input floor plans are converted to *.dxf format.

4.1 Setting up a working project in SAPFIR

Step 1. Open the **SAPFIR** program, create a new project, and configure the program environment. **Create – Project Properties** (Fig. 4.1, 4.2) – **Apply**. Leave the other settings at their default values.

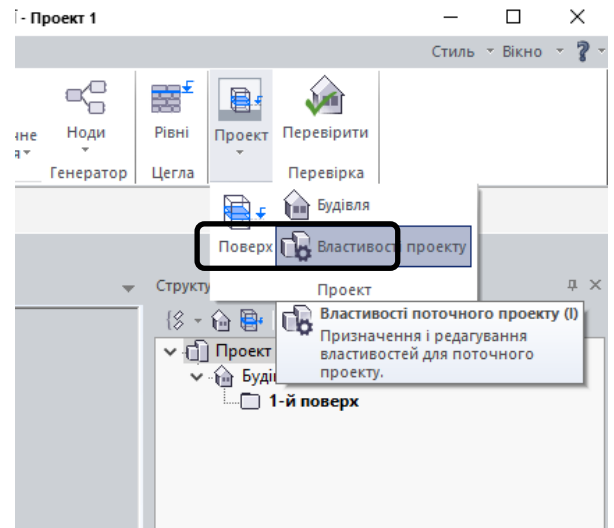
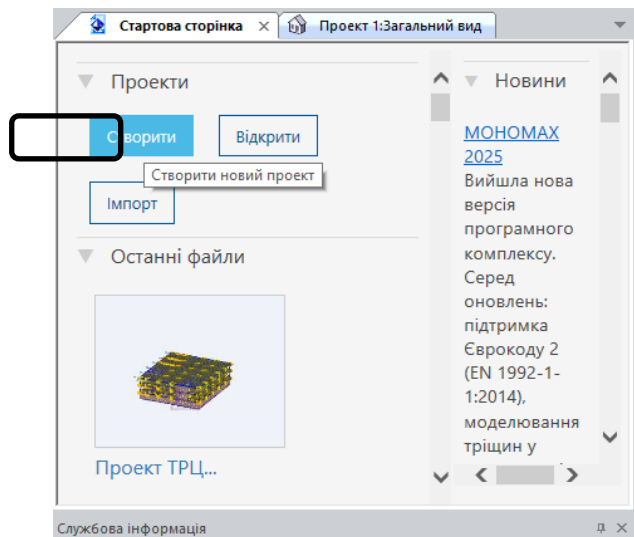


Fig. 4.1 Creating a new project

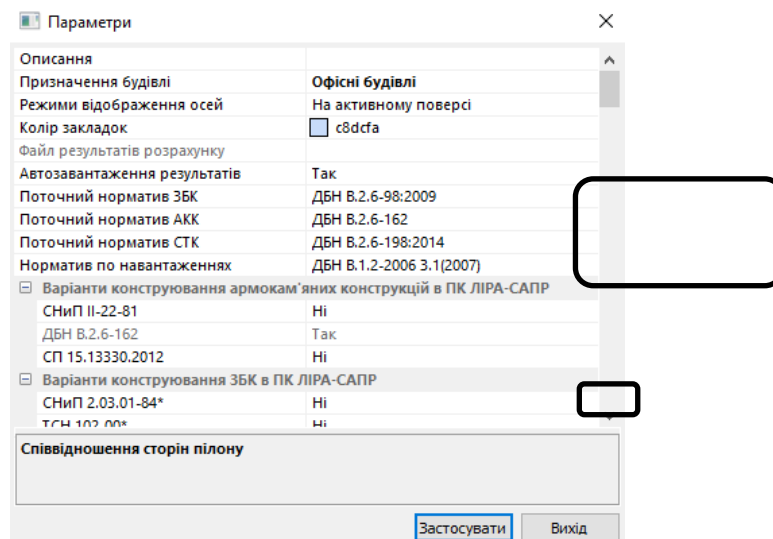





Fig. 4.2 Dialog window “Parameters”

Step 2. Visualization settings. Program menu  - **Settings – Visualization settings**. In the window, configure the metric grid parameters (**only in the first square**), number of cells – **40** (Fig. 4.3). - **Confirm**.

The “**Only in the 1st square**” grid is displayed only in the area with positive coordinates ($X>0, Y>0$), which is convenient for building models without ‘negative’ coordinates. “**Number of cells - 40**” determines how many grid squares are displayed relative to the origin. The larger the number, the larger the part of the workspace visible with the grid. This does not affect calculations or geometry, but it helps visually:

- ✓ to understand the elements in the model space more accurately;
- ✓ find construction points faster;
- ✓ avoid errors when elements are located “outside the model”;
- ✓ more conveniently check the orientation of axes and levels before exporting to **LIRA-CAD**.

Step 3. Save changes to the project. Program menu  - **Save as...**  - enter the name of the project, consisting of the author's surname, group number, and three digits of the version code (**Levkovsky_OM_164_281.**), specify the location where the project will be saved - **Save**.

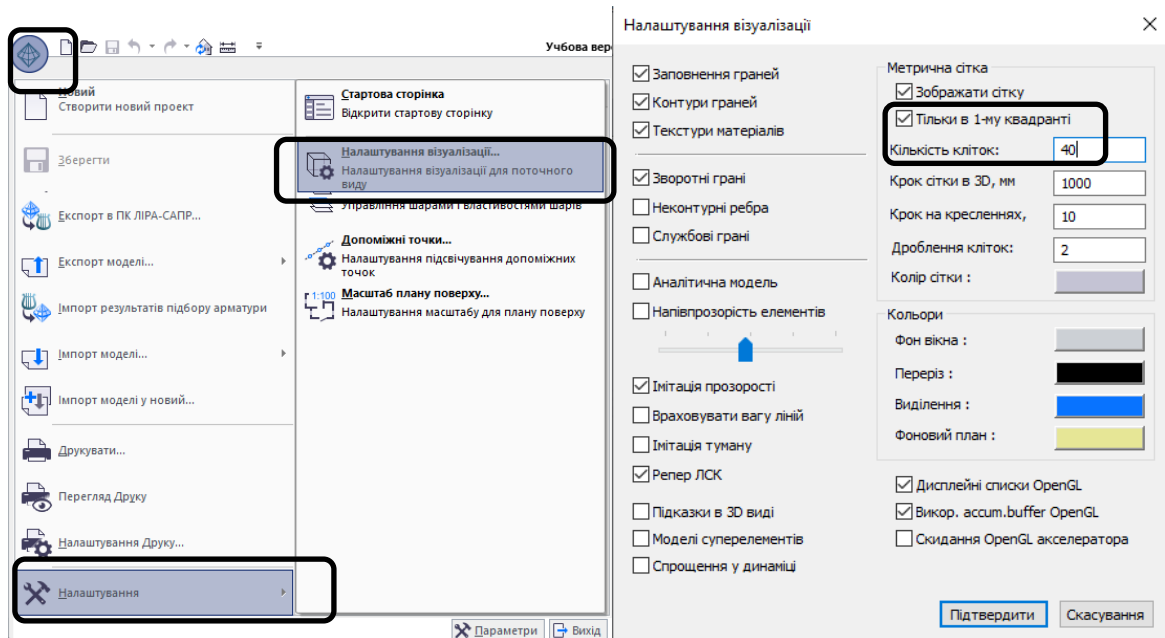



Fig. 4.3 Dialog window “Parameters”

4.2 Modeling the architecture of the “basement floor”

Step 1. Configure the floor properties. In the **Structure** window – **1st floor** – **Properties** – change the name “**Building 1**” to “**Office building**” and the floor name to “**Basement**” (Fig. 4.4). Set the floor height according to the task to 3650 mm and the floor bottom level to -3650 mm. – **Apply to object**  .

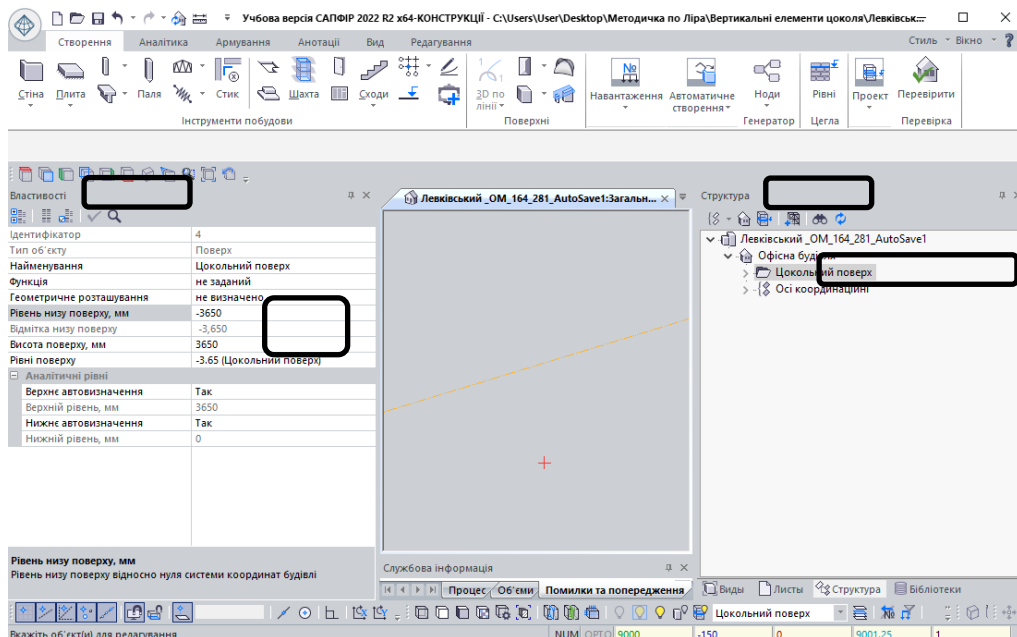




Fig. 4.4 Floor properties dialog window

Step 2. Create a coordinate axis system according to the project task. Tab **Create** – **Axes** – **Parameters** (Coordinate axes). Set up the numbering, Step of vertical and horizontal axes according to the option (Fig. 4.5) – **OK** – Link the axis grid to the coordinate system. Select the

axes with the cursor **Edit** – **Move**  - move the intersection point of axes “1”/“A” to the origin of the coordinate system. **Outline the dimensions**  - the dimensions are displayed on the coordinate axis grid.

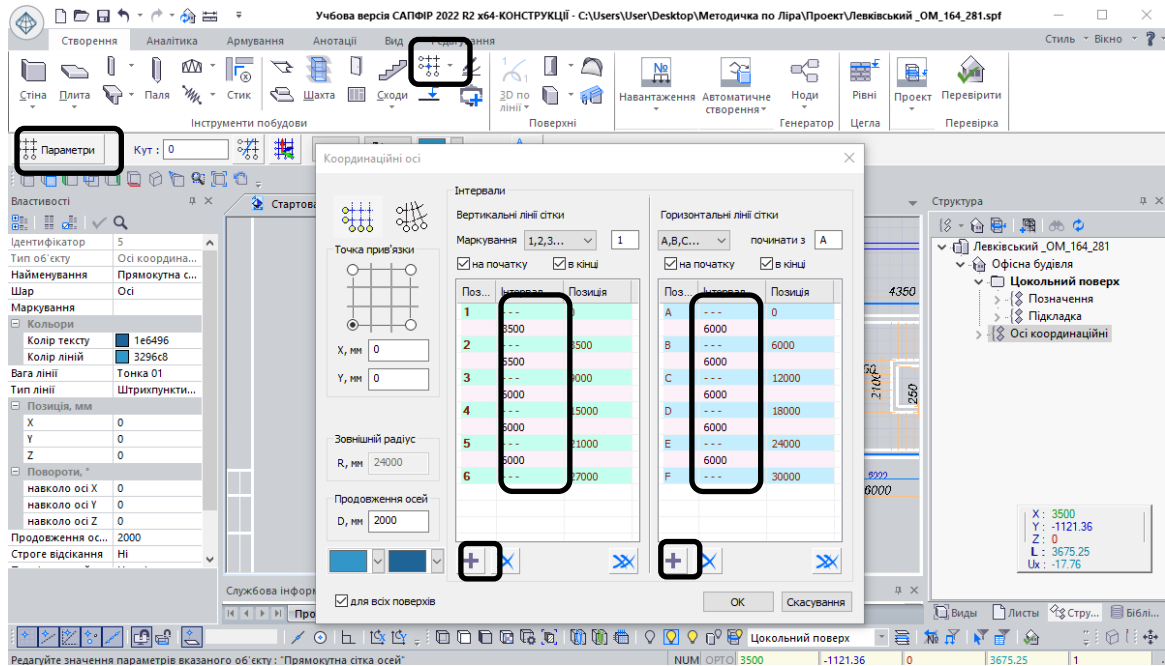


Fig. 4.5 Application window “Axes“

Step 3. Import the auxiliary “underlay” in *.dxf format. First, you need to develop a plan of the basement floor in *.dxf format. **Program menu** - **Import model... - AutoCAD drawing – Edit tab - Move** (Fig. 4.6) (move the “underlay” to the origin of the coordinate system, synchronize with the axis grid).

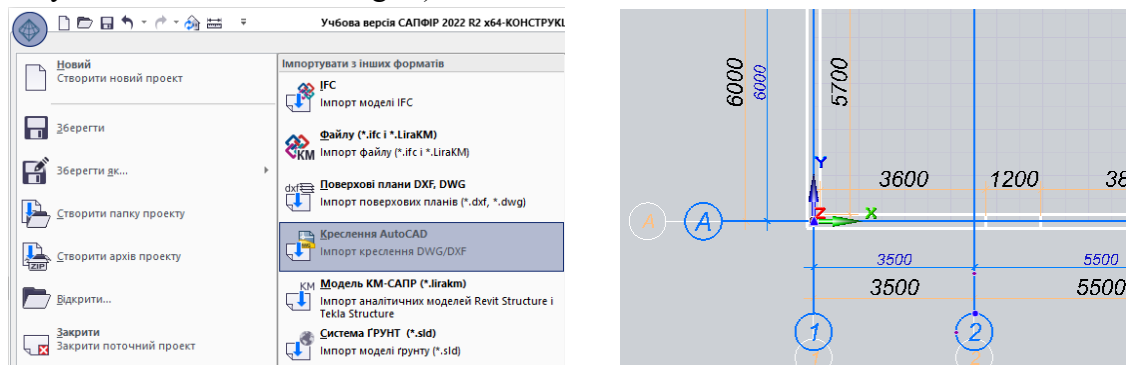
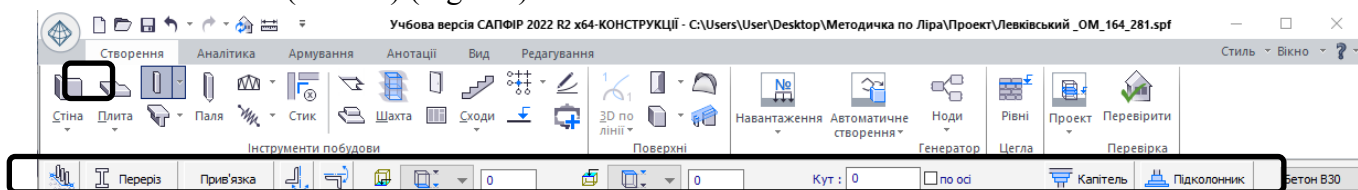


Fig. 4.6 Importing the basement floor “substrate.”

Step 4. Create columns. According to the task, columns measuring 400x400 mm are located in the central part of the building, made of C25/30 concrete. Go to the **Create tab – Column** – In the dialog box, the column properties are displayed (Cross-section, reference, height reference type, etc.). **Cross-section - Cross-section parameters** (assign a rectangular cross-section of 400x400 mm to the column) - **OK**. Place the columns according to the “underlay”. **Structure - right-click** on the name Column - **Select** (select all columns). In the Properties of Objects **dialog box - Material** - concrete B30 (C25/30) (Fig. 4.7).



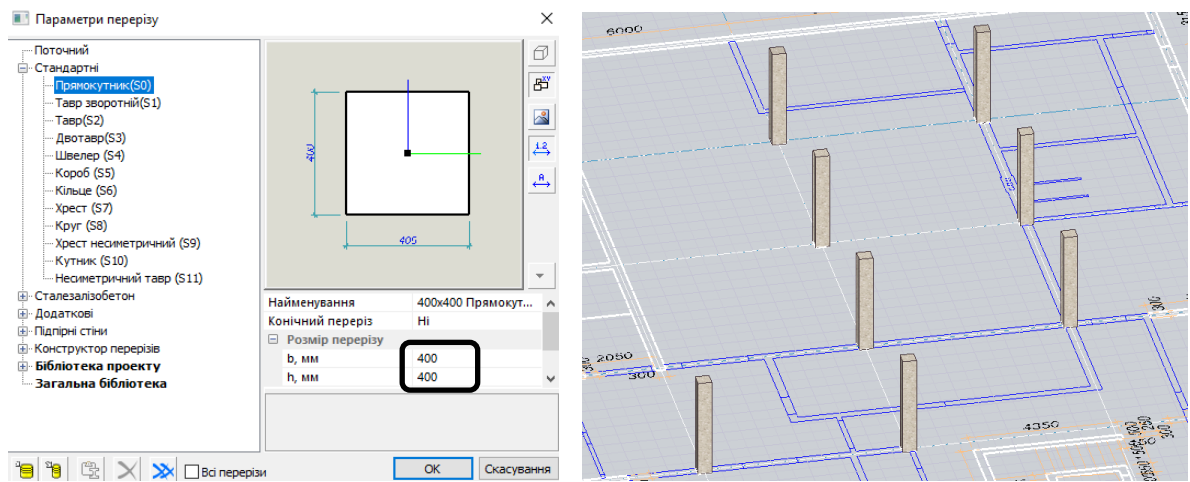



Fig. 4.7 Column cross-section parameters

Step 5. We create load-bearing walls with a thickness of 300, 250 mm and stiffening ribs of 400 mm, focusing on the “substrate,” material - concrete C20/25. The **Create** tab – **Load-bearing wall** – In the dialog box, the wall properties are displayed (thickness, reference, height reference type, etc.). We set the appropriate wall thickness and outline the “substrate.” Using the **Angle Cut** function , the **editing panels** combine two perpendicular parts of the wall (Fig. 4.8).

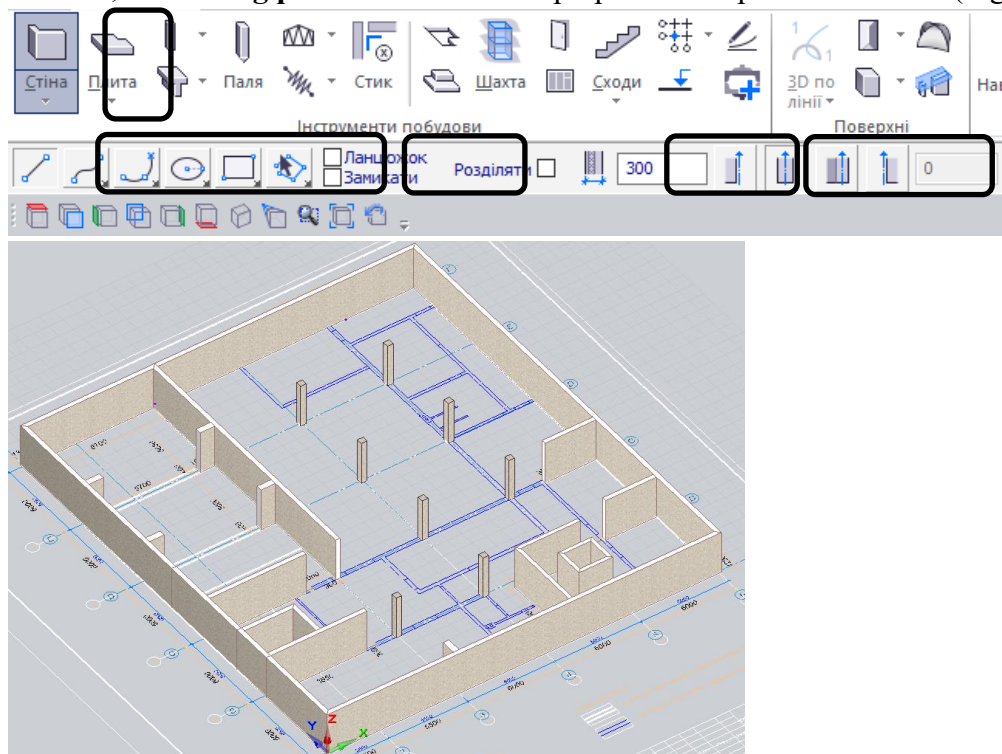


Fig. 4.8 Location of columns and walls on the basement floor

Step 6. Change the wall material to C20/25 concrete. In the project tree (**Structure**), select all walls – **right-click** on **Load-bearing walls** – **Select**, then the **Properties** window – Material concrete B25 (C20/25) (Fig. 4.9).

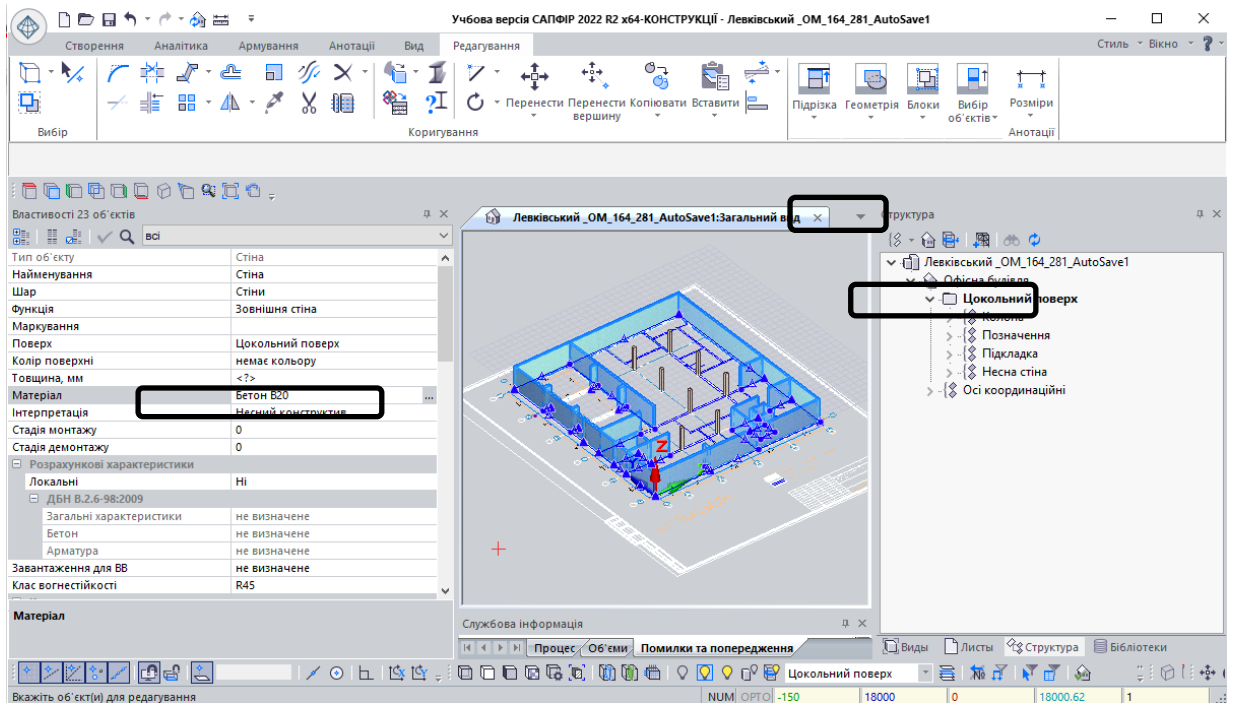


Fig. 4.9 Designation of material for vertical frame elements.

Step 7. Add windows. Windows are 800 mm high, according to the option, the bottom of the window is located at a mark of $-1.100\text{ m} = -(0.3\text{ m} + 0.8\text{ m})$. On the **Visualization – Frame** panel, make sure that the outline of the “lining” is visible. **Creation** tab – **Window** (set the window dimensions and its binding) (Fig. 4.10).

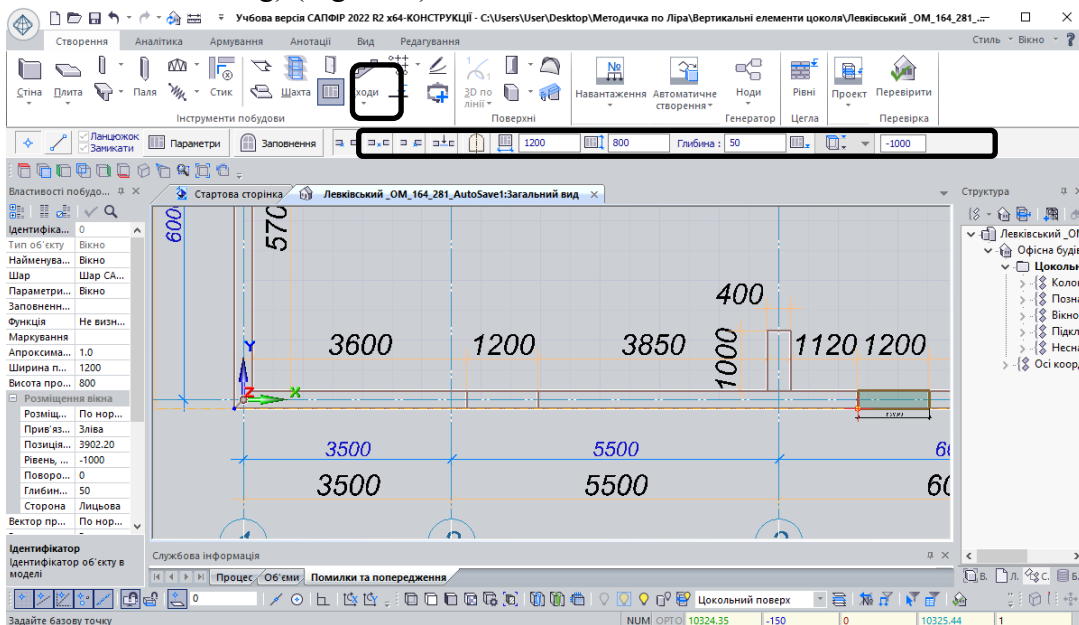


Fig. 4.10 Installation of window openings in load-bearing walls

Step 8. Add doors and gates. Door height is 2100 mm. Insert **Creation – Doors** (determine the door width according to the “underlay”). Make the gate height equal to the entire floor height of 3650 mm, since a beam must be specified above the gate (Fig. 4.11).

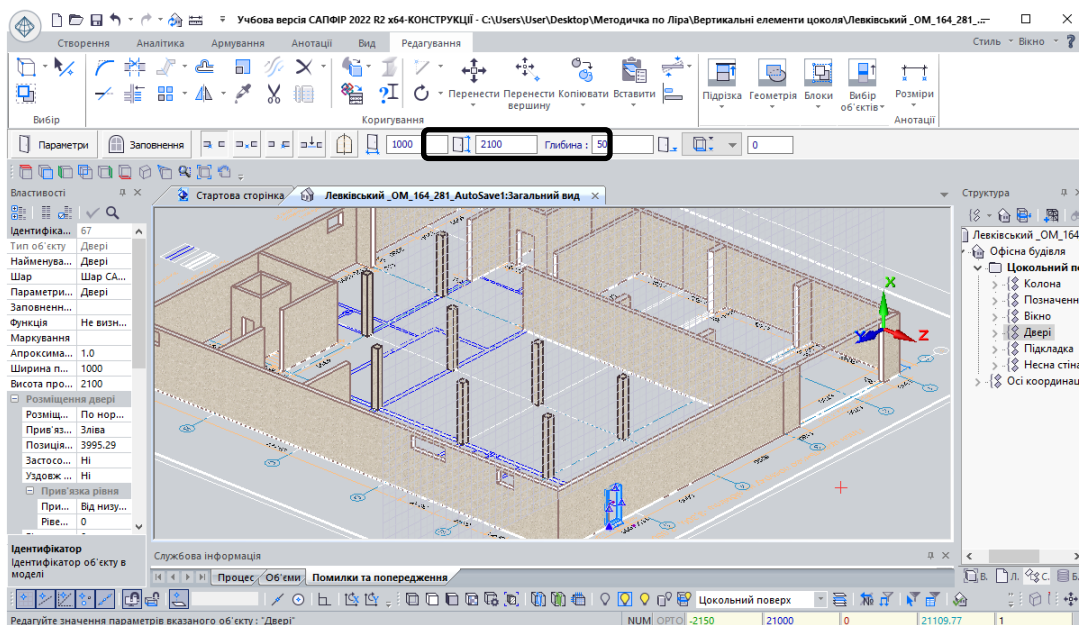


Fig. 4.11 Purpose of a doorway

Step 9. Add a foundation slab with a thickness of 350 mm. Insert **Create – Slab – Foundation slab**. Menu **Views – Top view**. Draw the outline of the slab on the “underlay.” Select the slab – **right-click** on **Foundation slab – Select**, then the **Properties** window – **Material** concrete B30 (C25/30) (Fig. 4.12).

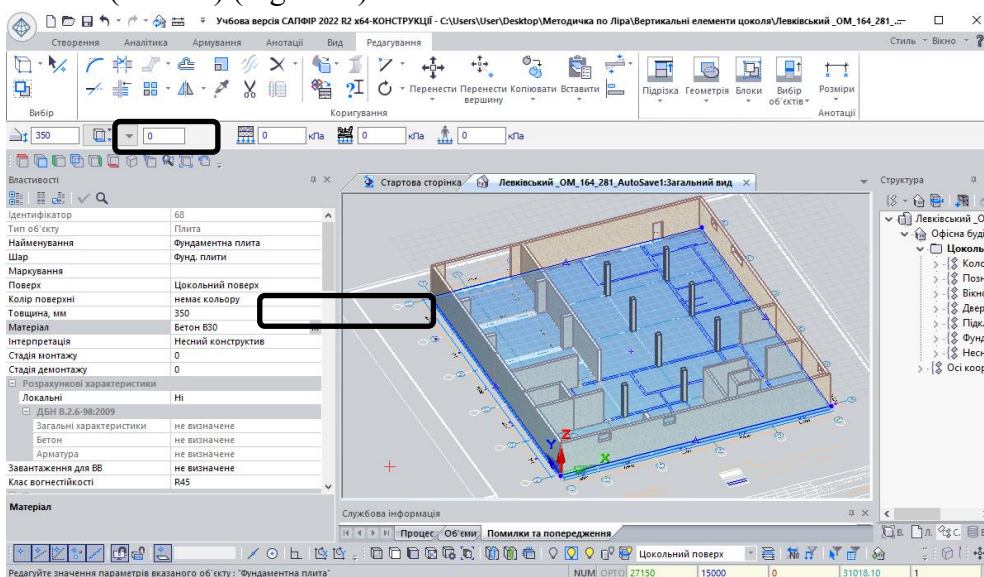



Fig. 4.12 Creation of a foundation slab

Step 10. We create internal partitions with a thickness of 300, 200, 120, 100 mm, focusing on the “lining,” the material is ceramic brick. Insert **Creation – Wall – Partition**. The wall properties (thickness, reference, height reference type, etc.) are displayed in the dialog box. In the **Wall Properties - Material** menu, select **Ceramic Brick**. Set the appropriate wall thickness and apply the walls to the “substrate.” Using the **Corner Cut**  function, the **editing panels** combine two perpendicular parts of the wall. Partitions are temporary structural elements added to the model as a linearly distributed load, taking into account door openings (Fig. 4.13).

Step 11. Add doors. The height of all doors is 2100 mm according to the task. Insert **Creation – Doors** (determine the width of the doors according to the “underlay”). Apply the doors to the plan using references.

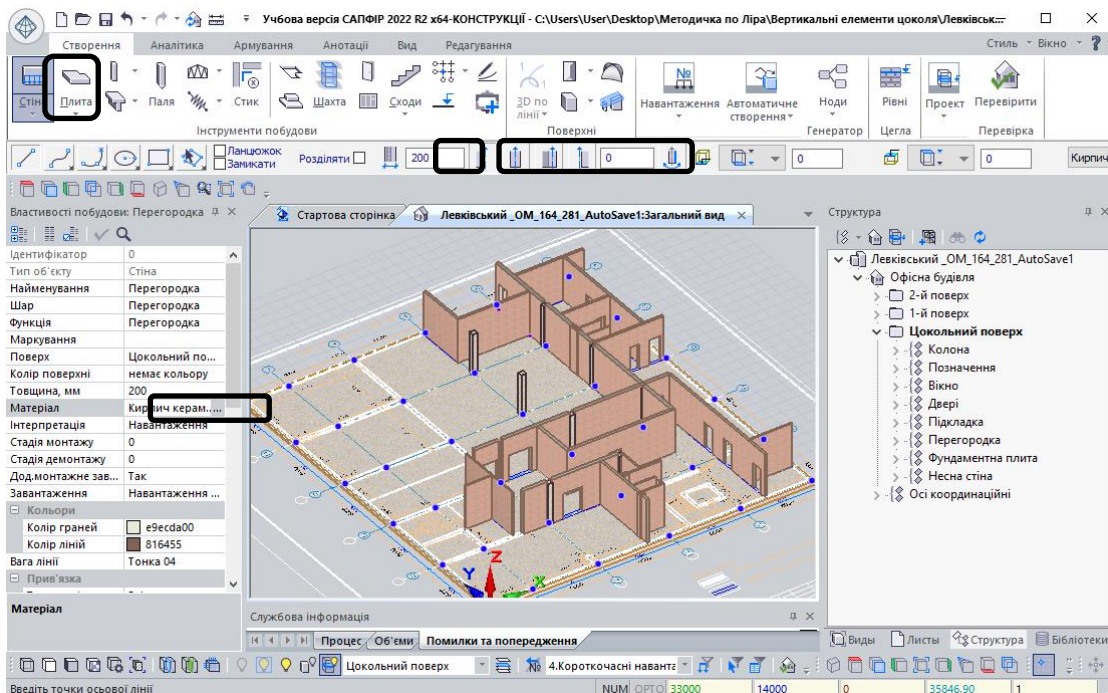


Fig. 4.13 Creation of internal partitions

4.3 Modeling the architecture of the “1st floor”

Step 1. Add the first floor to the project structure. In the **Structure** window, on the «*Office Building*» **right-click**, select **Create New Floor**. In the dialog box, change the floor name to “1st Floor” and set the **floor height** to 4500 mm. Click **OK** (Fig. 4.14).

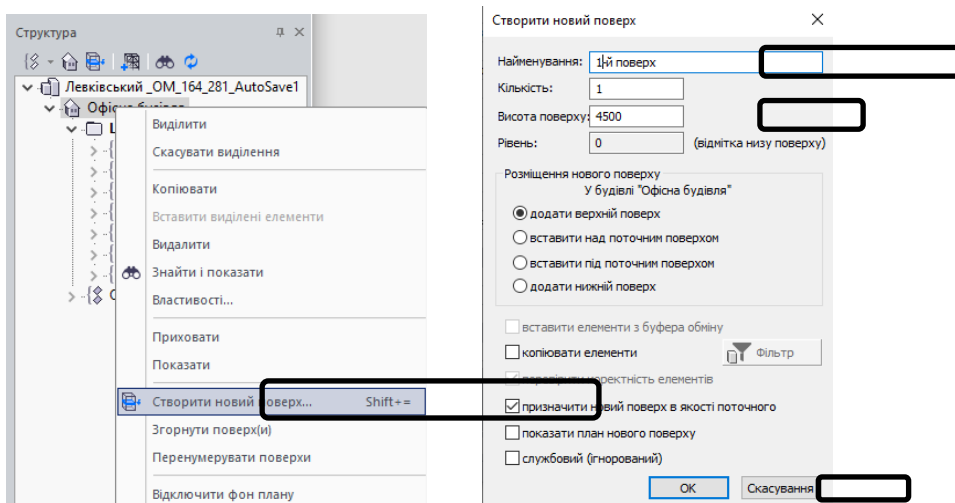



Fig. 4.14 Floor creation dialog box.

Step 2. In the Project **Structure**, twice **right-click** on the floor designation, we make the necessary floor active, it is highlighted in **bold**. We add the “underlay” of the 1st floor. Export the “base” in *.dxf format. **Program menu**  - **Import model... - AutoCAD drawing - Edit tab - Move** (move the “underlay” to the origin of the coordinate system, synchronize with the axis grid).

Step 3. For convenience, we hide the “lining” of the basement floor. **Structure – Basement floor, “underlay” – right click – Select – Visualization panel – Hide selected**. Similarly, you can hide individual structural elements and floors. To restore the overall appearance of the building, use the **Visualization Panel - Show All Elements** (Fig. 4.15).

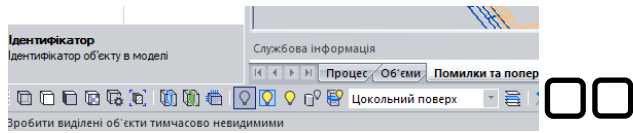


Fig. 4.15 Dialog box for displaying elements.

Step 4. We create columns on the 1st floor. First, make sure that the 1st floor is active. **Structure** – twice **left-click** on the 1st floor – highlighted in bold. According to the task, columns 400x400 and 300x300 mm are located in the central part of the building, material - concrete C25/30. The **Create** tab - **Column** - opens the column properties in the dialog box (Cross-section, reference, height reference type, etc.). **Cross-section** - **Cross-section parameters** (Fig. 4.16) (assign a rectangular cross-section of 400x400 mm and 300x300 mm to the column, respectively) - **OK**. Place the columns according to the “underlay”.

Step 5. We create the pylons for the first floor. According to the task, the pylons are 1200x300 mm (ratio 1/4), material - concrete C25/30. If the ratio is less than 1/4, the pylons are modeled using a bar finite element. If one side of the pile is more than 4 times larger than the other, the pile is modeled using plate finite elements.

Creation tab – **Column** – The column properties are displayed in the dialog box (Cross-section, reference, elevation reference type, etc.). – **Cross-section** – **Cross-section parameters** (assign a rectangular cross-section of 1200x300 mm to the column) – Set the desired reference (**Positional reference**) – **OK**. Place the columns according to the “underlay.”

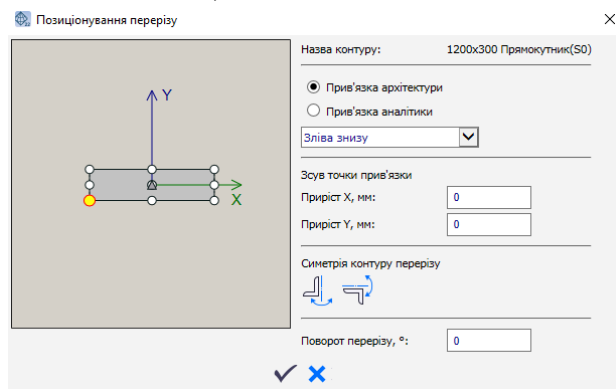



Fig. 4.16 “Cross-section positioning” dialog box

Step 6. We create load-bearing walls with a thickness of 300 and 250 mm, focusing on the “substrate,” material - concrete C20/25. The **Create** tab – **Load-bearing wall** – In the dialog box, the wall properties are displayed (thickness, reference, height reference type, etc.). We set the appropriate wall thickness and apply it to the plan, using the “ underlay” as a reference. Using the

Angle Cut function , the **editing panels** combine two perpendicular parts of the wall.

Step 7. Change the wall material to C20/25 concrete. In the project tree (**Structure**), select all **load-bearing walls** – **right-click** – **Select**, then the **Properties window** – **Material** concrete B25.

Step 8. Add doors to the elevator shaft, door height 2100 mm. Insert **Creation** – **Doors** (determine the door width according to the “underlay”). If necessary, use door binding to the plan.

Step 9. Add a 200 mm thick floor slab for the first floor. Hide the basement floor: **Structure** – click the **right-click** menu on the name “Basement floor” – Hide. **Create** tab – Slab, Views – Top view. Draw the slab outline on the “ underlay,” material C20/25 concrete. The slab has openings in the elevator shafts and for communications. **Creation** tab – **Cutout**, draw the outline of the corresponding openings on the slab (Fig. 4.17).

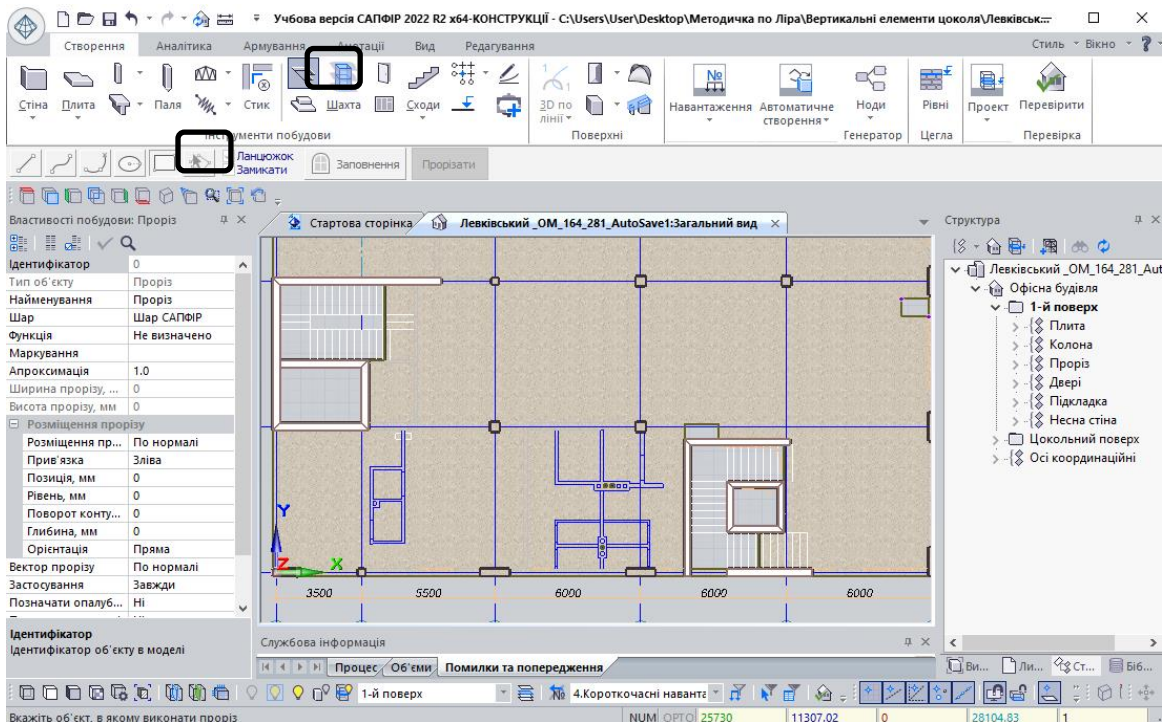


Fig. 4.17 Adding openings in the first floor slab.

Step 10. Add floor beams to the slab body. Beam 300x500h above the gate along axis “F” and 400x500h along axes ‘C’ and “D”. Insert **Create – Beam – Cross-section...** Create the appropriate beam cross-section. The beams are located in the slab body. The center of the beam is shifted down by half the height of the beam. **Reference point** – the top point of the beam. Apply the beams to the plan. Material – C20/25 concrete (Fig. 4.19).

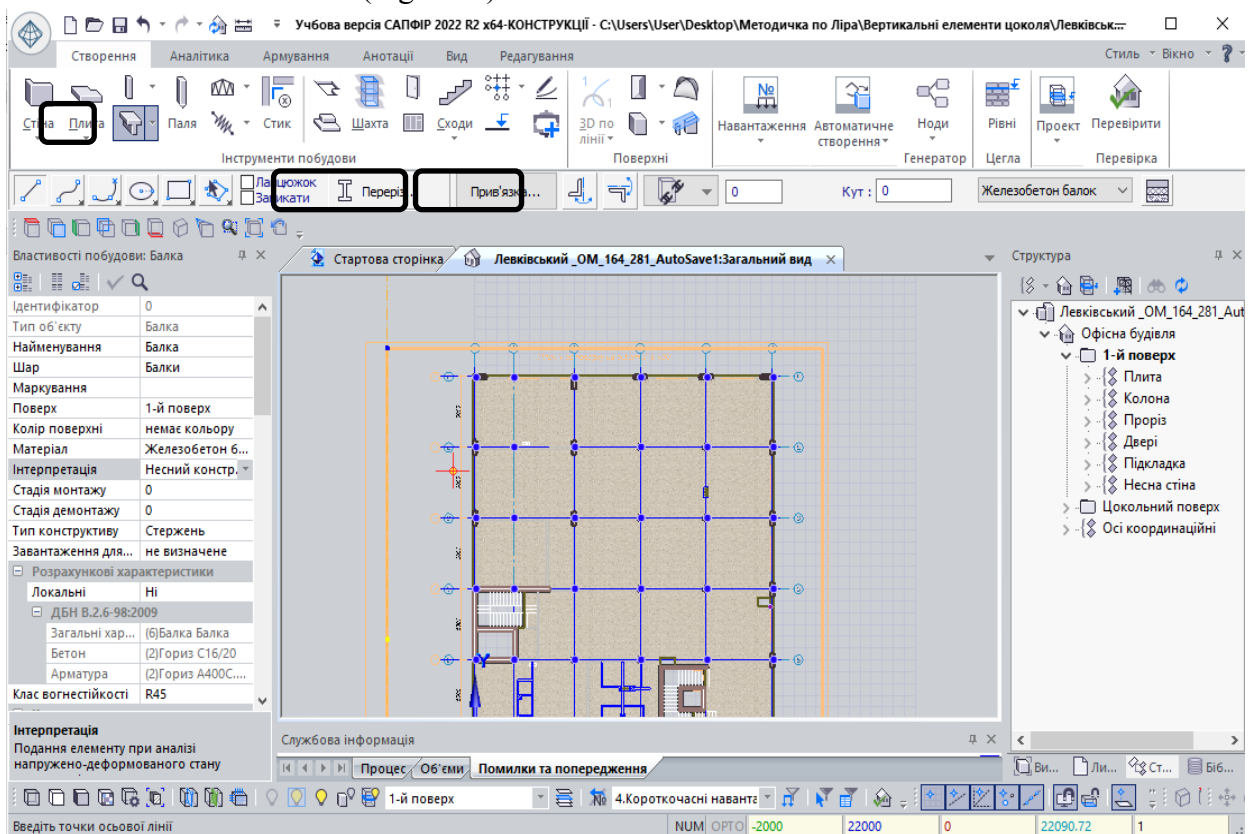


Fig. 4.18 Beam modeling

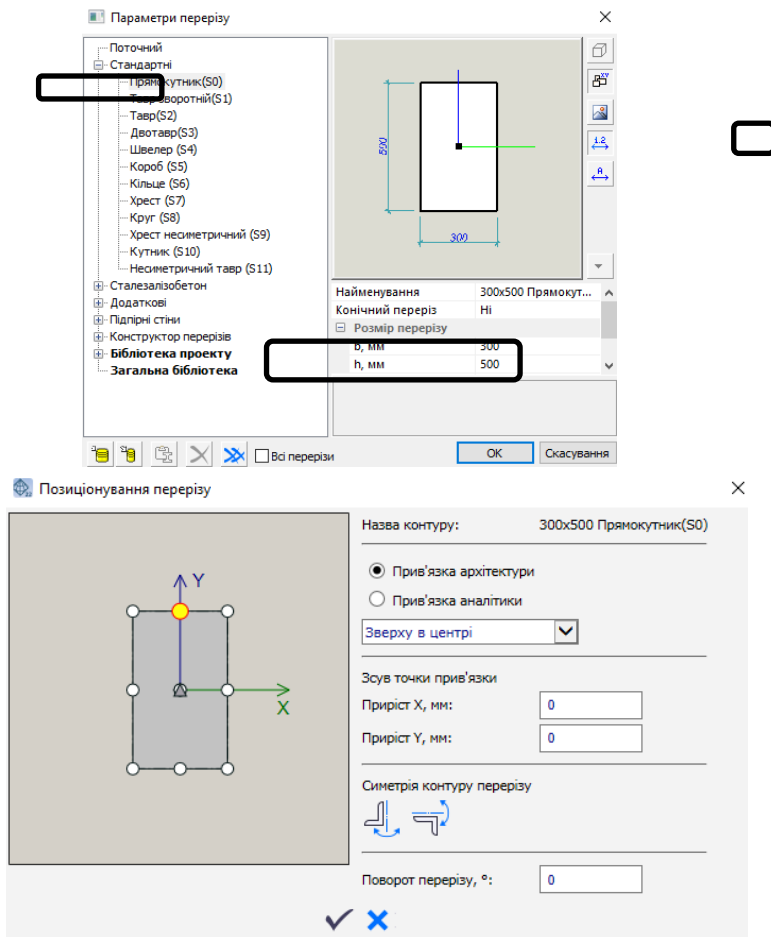


Fig. 4.19 Beam cross-section and attachment settings

Step 11. We create partitions with a thickness of 300, 200, 120, and 100 mm based on the “substrate.” The partition material is ceramic brick and D400 aerated concrete. D400 aerated concrete is not available in the material library. **Service - Settings - Material -** the “Materials” dialog menu opens - General tab - **right-click on Simple - Add material** (Fig. 4.20) - configure D400 aerated concrete according to its properties (Young's modulus - 2500 MPa, Poisson's ratio - 0.2, bulk weight - 400 kg/m³), **Interpretation - load**, in which case the walls will be perceived as a load. The algorithm for creating walls is described in **section 4.2 (Step 10, 11)**.

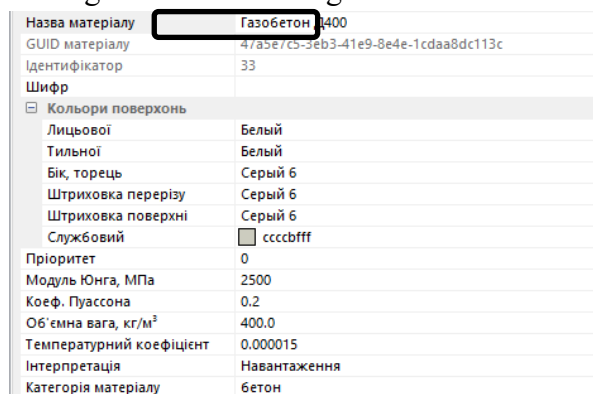


Fig. 4.20 Creating new material

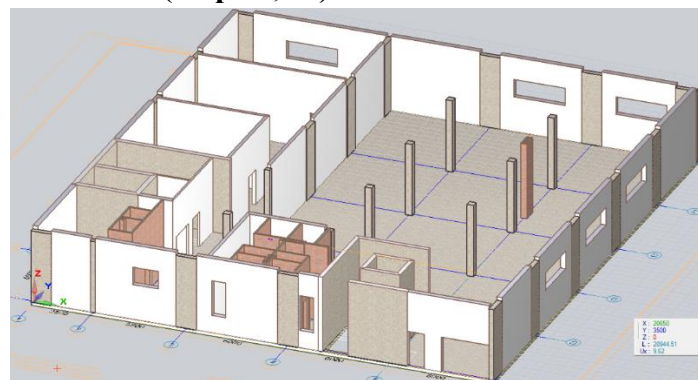


Fig. 4.21 Plan of the 1st floor

Step 12. The windows on the first floor are 1050 mm high, with the bottom of the window located at a height of 2010 mm. Between axes “A”/“3” and ‘A’/“4” there are windows with a height of 2100 mm, with the bottom of the window located at a mark of 1050 mm. The algorithm for creating walls is described in **section 4.2 (Steps 7, 8, 10, 11)**.

Step 13. We create the architectural structures of the second floor and the roof of the building in the same way (see **sections 4.2 and 4.3**).

5. ANALYTICAL MODEL OF THE BUILDING

The architectural model is converted into an analytical model using the **SAPFIR-CONSTRUCTIONS** subsystem. An important step is to check the correctness of the axes, align the plates and rods, and create a finite element mesh.

The following are generated automatically:

- ✓ bar elements (columns, beams);
- ✓ plate elements (floor slabs, walls);
- ✓ nodes and their connections
- ✓ material types (concrete, steel, wood), cross-sections according to the construction library and regulatory requirements are assigned.

5.1 Load assignment, modeling of interaction between elements

After creating a structural model, it is checked for:

- ✓ correct topology;
- ✓ connection of nodes;
- ✓ presence of all necessary properties of materials and cross-sections;
- ✓ absence of duplications or geometry errors.

Step 1. Sometimes in architectural models, columns do not intersect the floor slab, but only extend to its surface (Fig. 5.1). In this case, it is necessary to “extend” the column to the floor slab. Select the floor slab and the corresponding column - **Edit tab - Trim - Attach the base (or top) of**

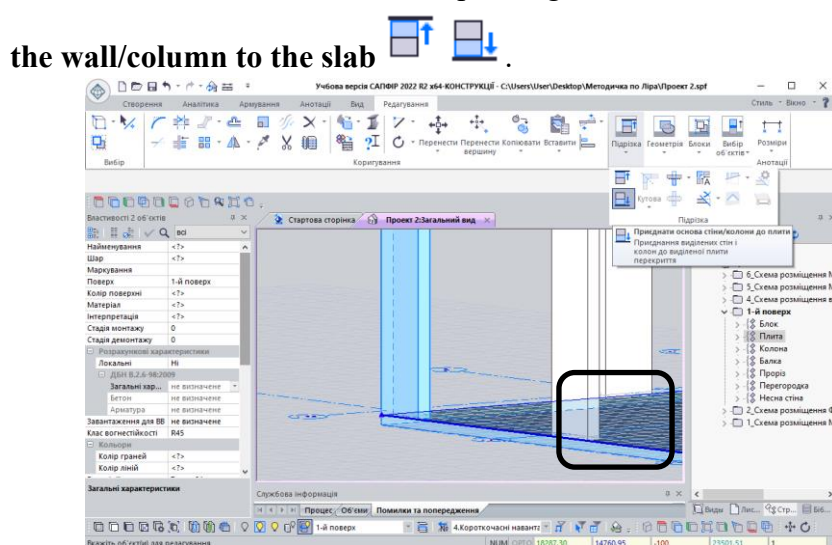


Fig. 5.1 Arrangement of slab and column intersection

Step 2. In our case, the beam in the floor slab has an offset center. When converting to an analytical model, the central axis of the beam does not pass through the middle plane of the plate, and the finite elements of the beam and slab do not intersect and are not connected to each other. Select the floor slab and all beams – **Properties – Analytical model parameters – Cross-section settings – Axial and volumes**. The program will understand that the beam and slab bodies are connected to each other. However, this function should only be used in isolated cases.

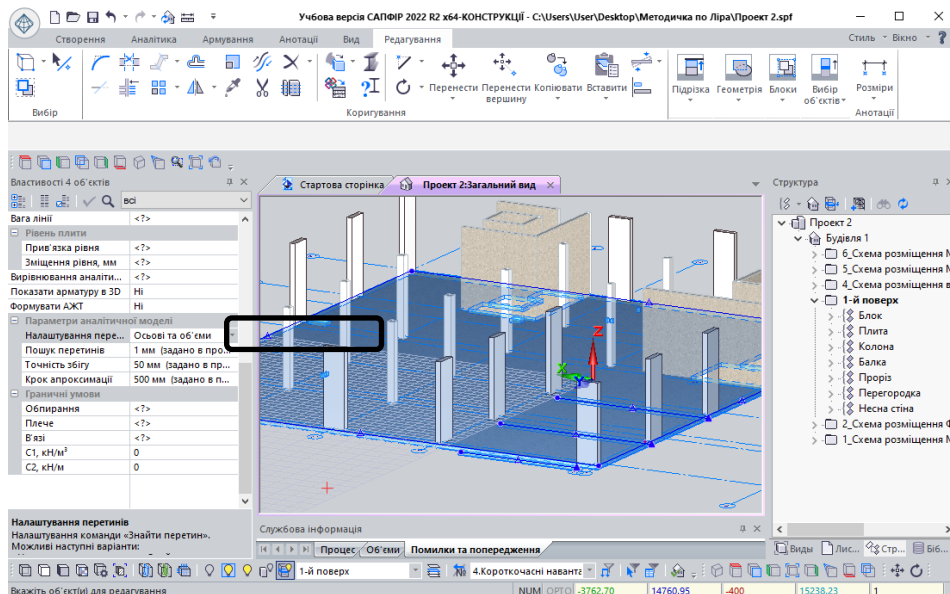


Fig. 5.2 Arrangement of slab and beam intersection

At the point where the column adjoins the floor slab, it is important to take into account the interaction of elements (**ARB** — analytical rigid body), because in the calculation model, the slab transfers its forces through only one node, while in the actual structure, the load is distributed over the contact area between the slab and the column. If the geometric thickness of the column is not taken into account in the model, this leads to a local concentration of forces and distortion of the stress-strain state. To increase the accuracy of the calculation in the joint area, rigid inserts or solid elements (**ARB**) are used to simulate the actual area of the slab's support on the column. This approach allows for adequate transfer of the load distribution, ensures the correct operation of the node, and avoids overestimating the stresses in the slab.

Step 3. Project structure – select all columns sequentially by floor – **Properties** – configure settings (see Fig. 5.3):

- **Material** – B30 concrete (all columns are made of C25/30=B30 concrete);
- **Interpretation** – load-bearing structure (a column is a load-bearing element of the frame);
- **Capital and Column base (triangulation of the support zone)** – Step of triangulation points 200 mm, rows of triangulation points 3, rows with fixed Step – 2 (create a finite element mesh refinement);
- **Extend** - Top and bottom (the rod element of the column extends to the middle plane of the slab);
- **Cross-section settings** - Axial non-rigid (In SAPFIR, the “**Axial non-rigid**” setting is used for columns to ensure correct connection to slabs and other elements even with small geometric deviations);
- **Form ARB** – Yes/

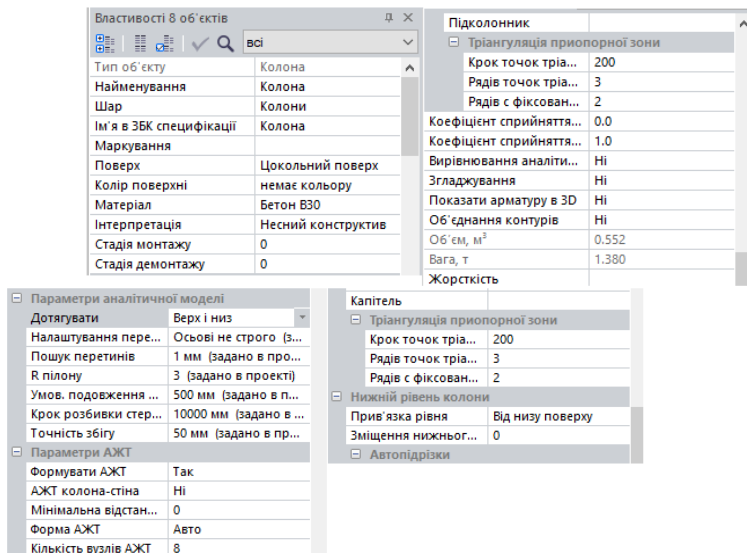


Fig. 5.3 Setting column properties

We calculate the load on the building frame. The **permanent** load from its own weight is added automatically, taking into account the density of materials (calculated or standard).

Step 4. We set the load on the floor (constant 1.8 kPa and continuous 4.8 kPa). Project **structure** – we select the floor slabs and assign the corresponding loads (Fig. 5.4).

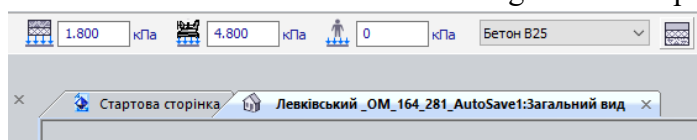


Fig. 5.4 Load assignment to floor slabs

Step 5. We apply loads to the roof slab (permanent load of 3 kPa, long-term load of 0.84 kPa, short-term snow load of 1.6 kPa). Project **structure**: we select the roof slabs and assign the corresponding loads (Fig. 5.5).

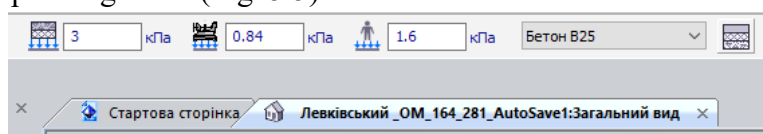


Fig. 5.5 Load assignment on the pavement

Step 6. In **SAPFIR**, wind loads are created using the “**Load** → **Wind**” command, where you specify the parameters of the wind area according to DBN, the height of the building, the direction of the wind, and the shape coefficients, after which the program automatically generates the wind pressure on the surface of the structure. The wind is applied along the X-axis and along the Y-axis (Fig. 5.6, 5.7).

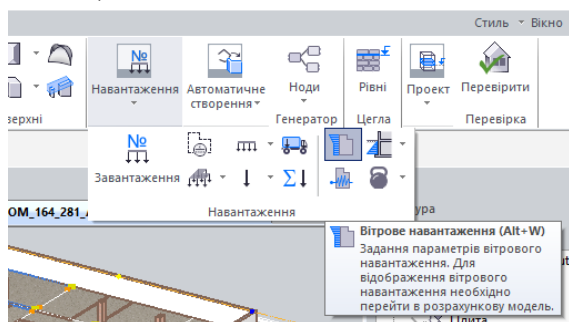



Fig. 5.6 Wind load design

Властивості	
Завантаження (наймен...	Вітер 0
Кут відн. ОХ, °	0
Рівень планування, м	0.0
Ширина будівлі, м	Авто
Спосіб прикладання	З - на стержневі елеме...
Реальні перерізи стер...	Ні
Архітектурна модель	Ні
Заморозити вітер	Ні
Нормативний документ	ДБН В.1.2-2006 З.1(2007)
Табличне задання	Ні
Параметри по ДБН В.1.2-2.2006	
Вітровий район Укр...	4
Вітровий тиск	0.055
Тип місцевості	I
Аеродинамічний ко...	1.4
Географічний коеф.	1.0
Динамічності коеф.	1.2
Надійності коеф. Yfm	1.14
Надійності коеф. Yfe	0.21

Властивості	
Завантаження (наймен...	Вітер 90
Кут відн. ОХ, °	90
Рівень планування, м	0.0
Ширина будівлі, м	Авто
Спосіб прикладання	З - на стержневі елеме...
Реальні перерізи стер...	Ні
Архітектурна модель	Ні
Заморозити вітер	Ні
Нормативний документ	ДБН В.1.2-2006 З.1(2007)
Табличне задання	Ні
Параметри по ДБН В.1.2-2.2006	
Вітровий район Укр...	4
Вітровий тиск	0.055
Тип місцевості	I
Аеродинамічний ко...	1.4
Географічний коеф.	1.0
Динамічності коеф.	1.2
Надійності коеф. Yfm	1.14
Надійності коеф. Yfe	0.21

Fig. 5.7 Setting wind load properties.

5.2 Verification and configuration of the analytical model

Step 1. Creation tab - Check  **Перевірити** - Check. Errors found. **AutoCorrect** - in the **Correction Selection** window, select the structural elements one by one and correct the errors indicated in the table (Fig. 5.8).

Перевірка моделі			
Елементи:			Ситуації:
Ім'я	Кільк.	Помил.	
<input checked="" type="checkbox"/> Колона	62	62	<input checked="" type="checkbox"/> Співпадіння поверхів
<input checked="" type="checkbox"/> Балка	9	3	<input checked="" type="checkbox"/> Зазор з поверхом вище
<input checked="" type="checkbox"/> Стіна	309	35	<input checked="" type="checkbox"/> Перетинання з поверхом вище
<input checked="" type="checkbox"/> Плита	4	4	<input checked="" type="checkbox"/> В однойменних блоках різна кількість елементів
<input checked="" type="checkbox"/> Двері	58	0	<input checked="" type="checkbox"/> В однойменних блоках не співпадають типи елементів
<input checked="" type="checkbox"/> Вікно	36	1	<input checked="" type="checkbox"/> Нульові розміри
<input checked="" type="checkbox"/> Проріз	62	45	<input checked="" type="checkbox"/> Збіг елементів
<input checked="" type="checkbox"/> Од	1	0	<input checked="" type="checkbox"/> Од. Співпадіння на різних рівнях з елементом
			<input checked="" type="checkbox"/> Перетинання з елементом
			<input checked="" type="checkbox"/> Включає елемент
			<input checked="" type="checkbox"/> Матеріал не визначений
			<input checked="" type="checkbox"/> Кількість несучих шарів більше одного
			<input checked="" type="checkbox"/> Товщина несучого шару дорівнює нулю

Елементів, що перевіряються: 541

Перевірка Вихід

Автокорекція

Для припинення перевірки натисніть клавішу Esc

Fig. 5.8 Model verification

Let's move on to checking the correctness of the analytical model. Go to the **Analytics tab** – **Calculation Model** – **Regenerate Calculation Model** – check the box to tighten twice and intersection – **OK**. **Analytics** – **Network** – **Create Triangulation Network** (Fig. 5.9).

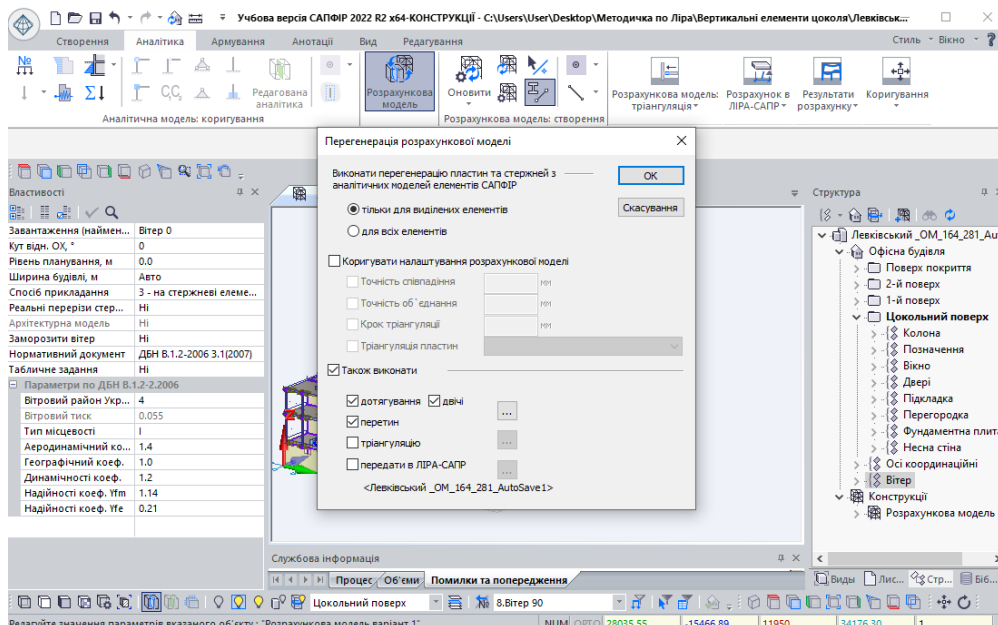


Fig. 5.9 Configuring the properties of the analytical model construction.

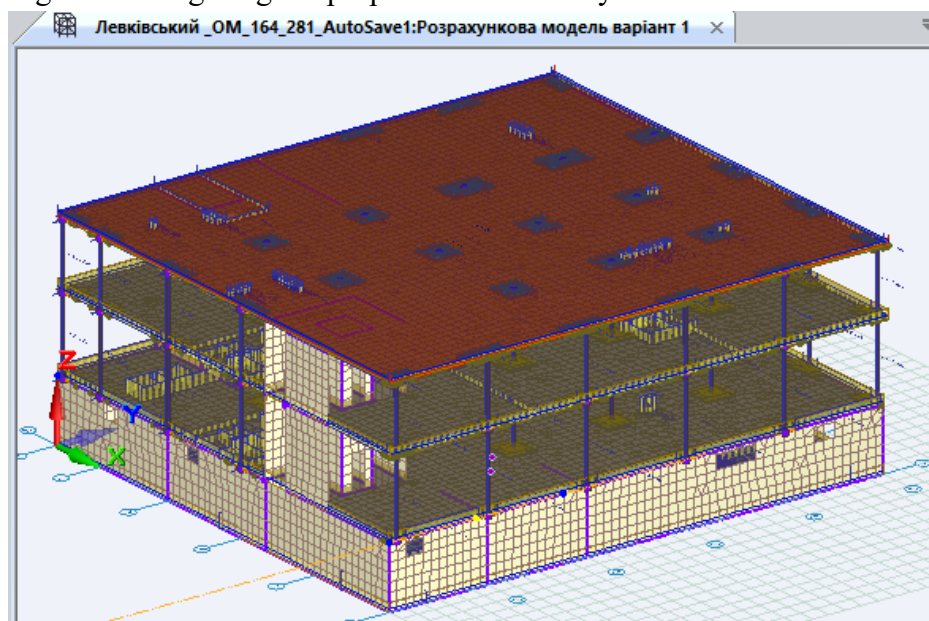


Fig. 5.10 Analytical model of a building

Step 2. Perform a visual inspection of the intersections of the main structural elements. In the Analytical Model, these areas are highlighted in blue (Fig. 5.12). First, hide the load visualization. Go to the **View** tab, select **Modeling Layers**, and disable load visualization (Fig. 5.11).

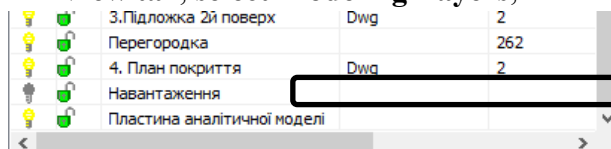


Fig. 5.11 Disabling load visualization

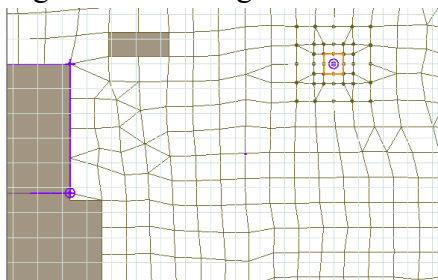


Fig. 5.12 Visualization of intersections of structural elements. Floor slab – wall, floor slab – column. Column-slab ARB is shown.

If all structural elements are connected to each other by joints, proceed to the calculation in **LIRA-CAD**.

Step 3. Analytics – Open – Launch VIZOR-CAD for static and dynamic calculation and selection of reinforcement. After that, **LIRA-CAD** opens with an analytical model of the building (Fig. 5.13). Save the project in a separate file.

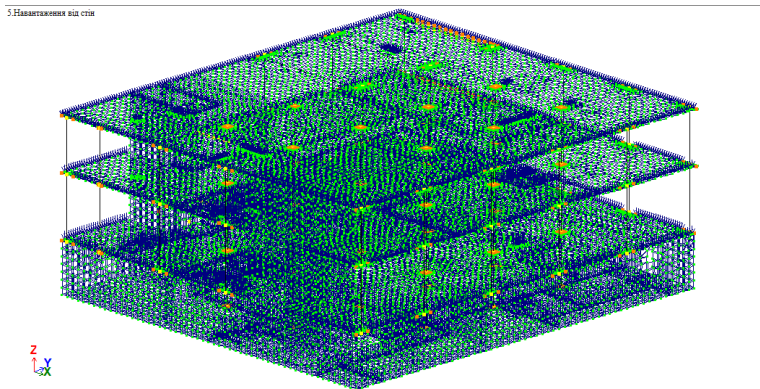


Fig. 5.13 Analytical model of a building in **LIRA-CAD** software.

After creating an architectural and structural model in the **SAPFIR** environment and transferring it to the **LIRA-CAD** calculation system, the user obtains the basis for a spatial calculation scheme. Geometric elements (nodes, rods, slabs), materials, loads, and connections are automatically generated in the model. However, for the calculation to be performed correctly, additional configuration and refinement of parameters in the **LIRA-CAD** environment is required.

The **LIRA-FEM** software package implements the classical **finite element method (FEM)**, which consists in discretizing a structure into separate elements—beams, slabs, shells, or solid bodies connected at nodes. Each element is characterized by its own geometric, stiffness, and material parameters.

The main idea behind MCE is that instead of solving complex equations for the entire structure at once, the problem is broken down into a set of local equations for each element, which are then combined into a **global system of equilibrium equations**.

The method allows determining:

- stress-strain state of structures;
- displacements, forces, and reactions in supports;
- stress concentration zones and possible failure locations.

Thanks to its versatility and high accuracy, MCE has become the primary method in modern engineering systems for structural calculations, particularly in **LIRA-FEM**.

As part of **BIM** technology, an architectural model created in **SAPFIR** contains geometric, material, and informational components for each element of a building. However, for engineering calculations, it is necessary to form an **analytical model** that simplifies the geometry to the axes and surfaces of the elements but preserves all physical and mechanical properties.

When exporting data from **SAPFIR** to **LIRA-FEM**, the following occurs:

- automatic creation of a finite element mesh;
- transfer of material parameters (modulus of elasticity, density, concrete class, reinforcement type);
- boundary conditions are formed (clamping, hinges, supports);
- coordinates and load systems are coordinated.

This ensures **information compatibility** between architectural modeling and computational analysis.

6. CONCLUSIONS

Engineering design of modern buildings is impossible without creating an accurate computational model that reflects the structural features of the object, the properties of materials, and the action of external loads. This task is extremely important in the educational process, as it allows students to combine their knowledge of structural mechanics, strength of materials, and computer modeling.

The case study focuses on creating an **information model of a multi-story office building** in the **SAPFIR** environment, followed by transferring it to **LIRA-FEM** for analytical calculation. The task simulates the real work process of a designer, where it is necessary to take into account the architectural structure, materials, loads, and boundary conditions that affect the behavior of structures.

Subsequently, the analytical model is refined in the **LIRA-CAD** software package and calculations are performed. This allows students not only to verify the correctness of the model construction, but also to evaluate the behavior of the building under real loads.

The results of the analysis demonstrate:

- the influence of the type of supports and the stiffness of elements on stress distribution;
- the importance of correctly specifying material characteristics;
- the need to control the topology of the analytical model.

The analysis of the situation in the **LIRA-FEM** environment is the final stage of the case study, which combines students' theoretical knowledge with practical digital calculations.

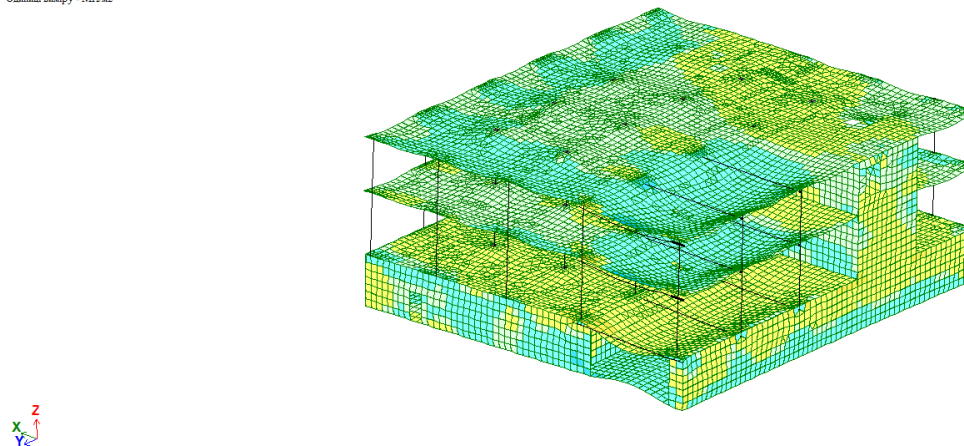
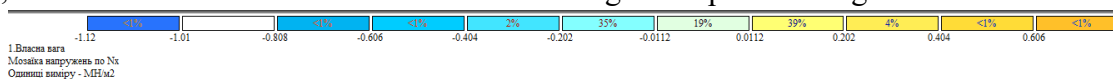


Fig. 5.14 – Example of isopoles of normal stresses

7. QUESTIONS FOR DISCUSSION

1. What is the difference between architectural and analytical building models?
2. Why is it necessary to check the topology of element connections when switching from SAPFIR to LIRA-FEM?
3. What are the advantages of using the BIM approach compared to traditional 2D design?
4. How does the finite element method allow you to evaluate the stress-strain state of structures?
5. Which stages of building an analytical model are considered most critical for the correctness of the calculation?
6. What errors can occur when forming a finite element mesh, and how can they be avoided?

7. Which parameters are most often adjusted after preliminary calculation in LIRA-FEM?
8. Can a BIM model replace a design engineer, or does it only help to increase the efficiency of their work?
9. How does digital design change the role of an engineer in a team?
10. How does the integration of architectural and computational modeling affect the quality of projects and the responsibility of specialists?
11. What skills will be most in demand for engineers working with BIM?
12. How can the knowledge gained in this case study be used in a thesis or research project?

8. LIST OF REFERENCES USED

1. Edited by Prof. V.V. Boichenko, Doctor of Technical Sciences, SAPFIR 2022. Textbook. Medvedenko D.V., Palienko O.I., Shut O.O. Edited by Dr. Tech. Sci., Prof. M.S. Barabash. Publisher: LIRALAND Group, 2022. 137 p.
2. Methodological user guide for the LIRA-SAPR 2022 software package. - Kyiv: LIRA Soft, 2022. - 456 p. (Official user guide containing a detailed description of the interface, principles of model building, calculations, and analysis of results in LIRA-FEM.)
3. LIRA-SAPR. Integration with SAPFIR: User Guide. - Kyiv: LIRA Soft, 2021. - 210 p. (Methodological recommendations for the two-way exchange of models between SAPFIR and LIRA-FEM, the creation of BIM projects, and the organization of a shared digital environment.)

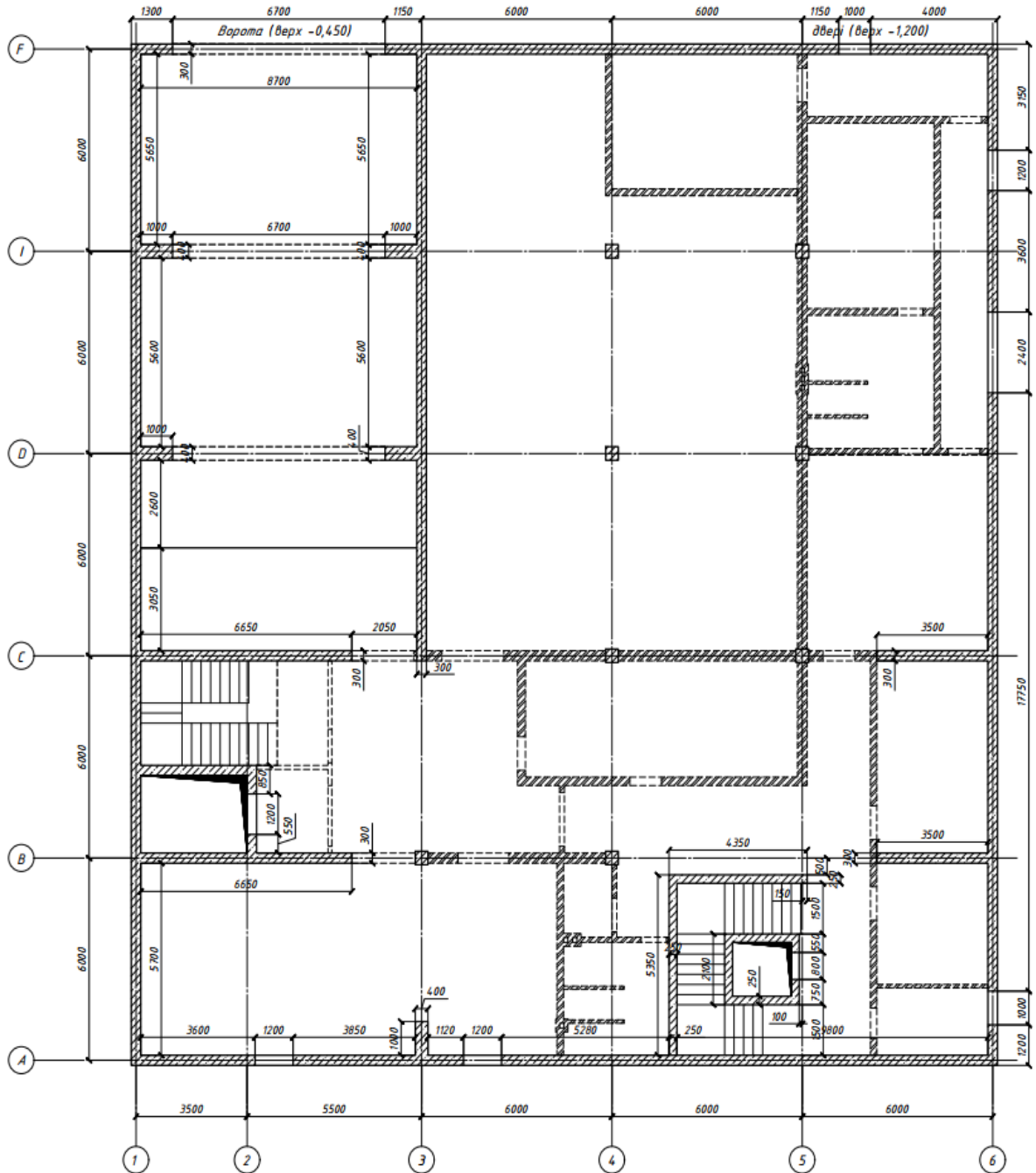
Electronic resources

1. <https://www.liraland.ua/>

APPENDIX A

Floor plans of the building



План цокольного поверху на відмітці -3,300,



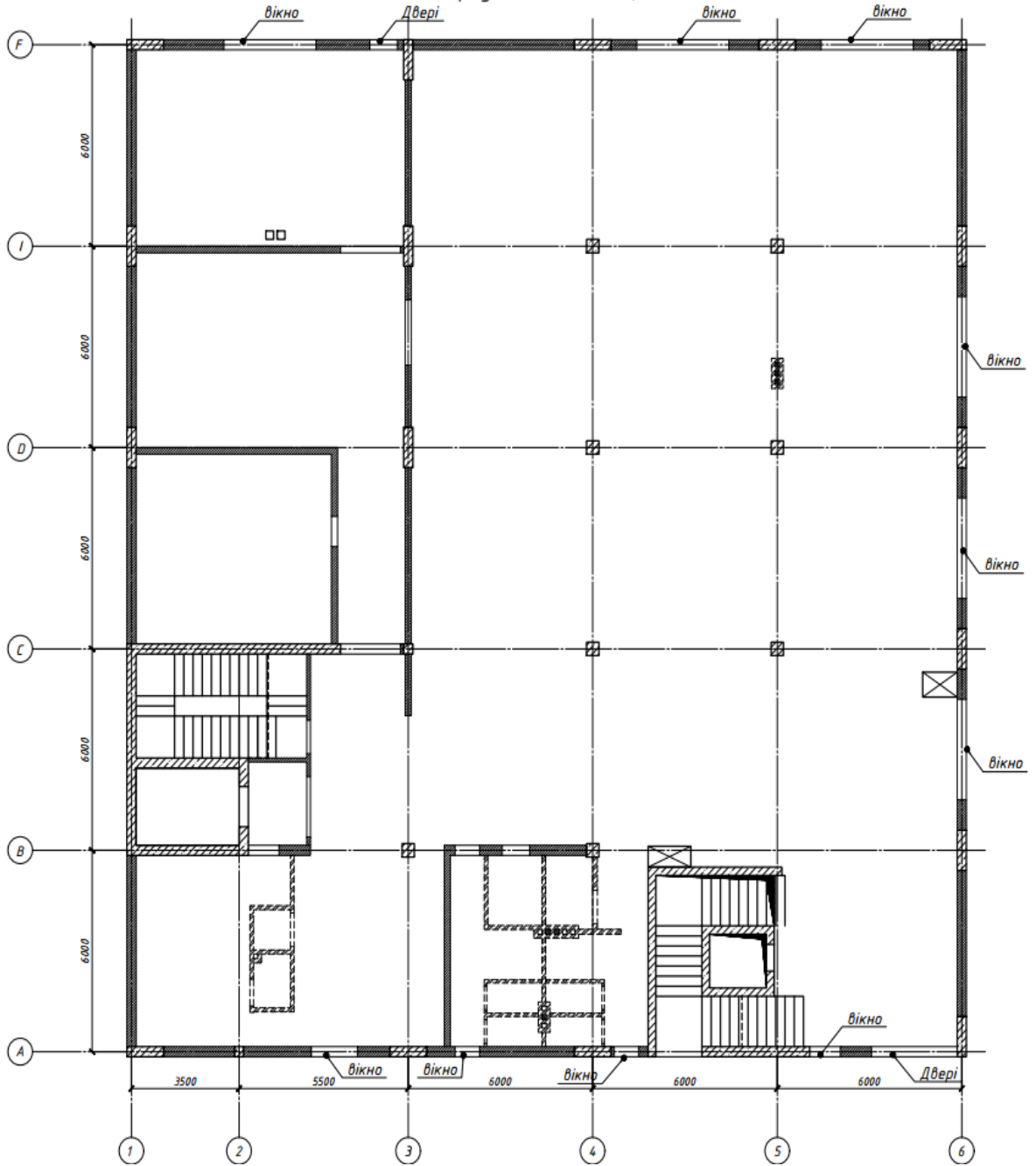
Примітка:

1. Всі вікна висотою 800 мм, верх вікон на відмітці -0,300;
2. Всі двері висотою 2100 мм;
3. Тимчасове навантаження на перекриття 400 кг/м²;
4. Вага складу перекриття (плитка 20 мм, стяжка 80 мм);
5. Всі колони перерізом 400x400 мм.

Умовні позначення:





-  - монолітні вертикальні елементи стіни, колони;
-  - внутрішні перегородки, керамічна цегла;

План 1-го поверху на відмітці 0,000

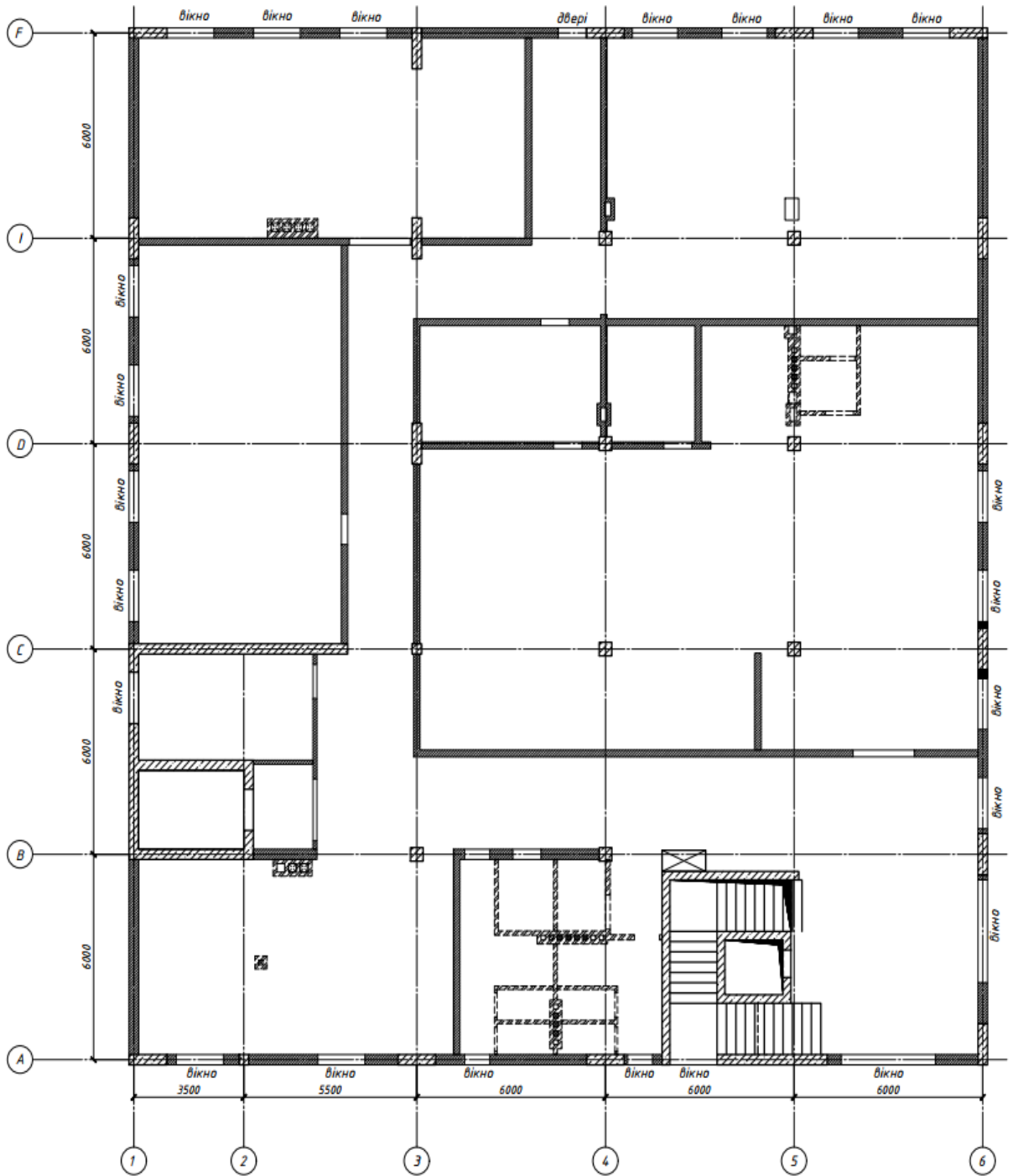


Примітка:
 1. Всі вікна висотою 1050 мм, верх вікон на відмітці 3,150;
 2. Всі двері висотою 2100 мм;
 3. Тимчасове навантаження на перекриття 400 кг/м²;
 4. Вага складу перекриття (плитка 20 мм, стяжка 80 мм);
 5. Ролони перерізом 400x400 мм та 300x300 мм.





Умовні позначення:

-  - монолітні вертикальні елементи стіни, колони;
-  - внутрішні перегородки, керамічна цегла;
-  - перегородки, зовнішні стіни газобетон 200 мм та 300 мм;
-  - позначення отвору в перекритті між цоколем та 1-м поверхом.

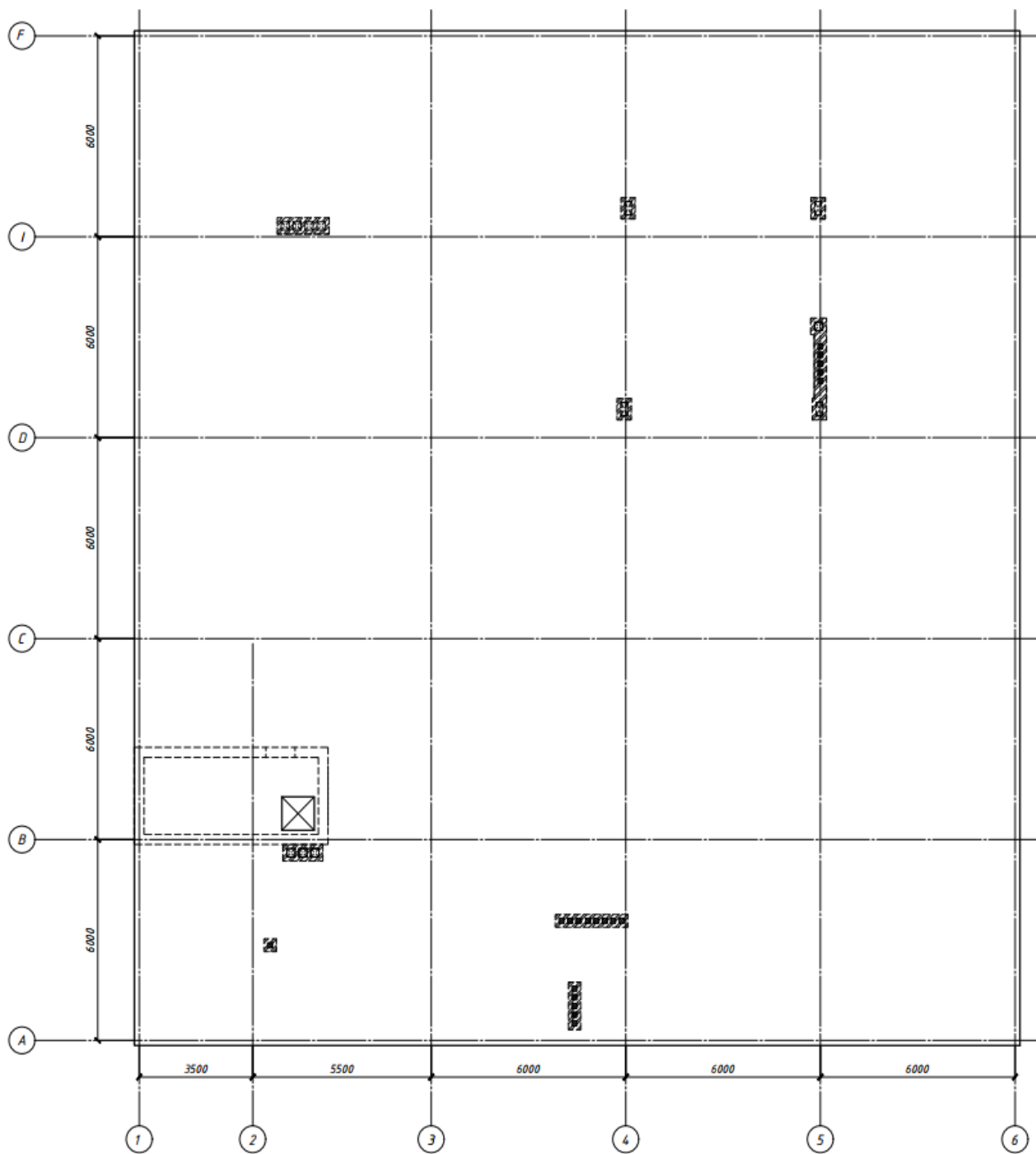
План 2-го поверху на відмітці +4,500



Примітка:
 1. Всі вікна висотою 2100 мм, верх вікон на відмітці +7,650;
 2. Всі двері висотою 2100 мм;
 3. Тимчасове навантаження на перекриття 400 кг/м²;
 4. Вага складу перекриття (плитка 20 мм, стяжка 80 мм);
 5. Колони перерізом 400х400 мм та 300х300 мм.

- Умовні позначення:
-  - монолітні вертикальні елементи стіни, колони;
 -  - внутрішні перегородки, керамічна цегла;
 -  - перегородки, зовнішні стіни газобетон 200 мм та 300 мм;
 -  - позначення отвору в перекритті між 1-м та 2-м поверхом.

План покриття на відмітці +8,300



9. MODELING OF SOIL-FOUNDATION INTERACTION WITH PLAXIS 3D SOFTWARE

Kovba V.V., Zahilskyi V. A., Siedin V. L.

1. Introduction

The integration of PLAXIS 3D software into the educational process allows students to acquire practical skills in using advanced digital technologies that meet the requirements of modern engineering practice.

PLAXIS 3D is a specialized software package designed for three-dimensional numerical modeling of soil masses, foundations, underground, hydraulic, and transportation structures using the finite element method [4, 5]. It is widely used in geotechnical design and scientific research to analyse the stress–strain state of foundations and structures [2], predict settlements, assess slope stability, evaluate the performance of pile foundations [8], and study the interaction between structures and the soil medium.

The PLAXIS 3D software package is one of the world’s leading tools for three-dimensional numerical modeling of soil masses, foundations, underground, hydraulic, and transportation structures based on the finite element method.

The aim of the course is not only to master the technical use of PLAXIS 3D, but to develop the ability to think critically and make engineering decisions based on the results of numerical analysis. Future professionals must learn to perform calculations related to slope stability, foundation settlements and displacements, interaction of pile systems with soil, as well as the behavior of retaining walls and deep excavations, and complex underground structures such as tunnels and mine workings.

The application of PLAXIS 3D provides the opportunity to study the stress–strain state of the soil foundation, taking into account the nonlinear behavior of the material [1], plastic deformations, rheological properties, and interaction with structural elements. Thus, students gain practical experience reflecting real engineering conditions and can help work effectively in design and research organizations in the future.

The **purpose** of integrating PLAXIS 3D into the training of future engineers is not only to familiarize them with the interface and tools of the software product, but also to develop comprehensive professional competencies in the field of geotechnical design.

The task is to prepare specialists who are able to:

- to create computational models of soil masses and structures in PLAXIS 3D;
- to model complex soil conditions and engineering structures in a three-dimensional environment;
- to apply modern numerical methods to solve geotechnical problems;
- to perform calculations in accordance with national regulations (DBN, DSTU) and European standards (Eurocode 7);
- to integrate modeling results into design and operation of buildings and structures;
- to critically evaluate the obtained results by comparing them with manual methods and simplified computational models;
- to perform calculations of slope stability, foundation settlements, interaction of pile systems with the foundation, and the behavior of retaining walls, excavations, and tunnels;
- to analyse the results of numerical modeling and interpret stress, strain, and displacement diagrams;
- to compare the results of numerical calculations with manual methods and the outcomes obtained from other software packages.

After completing the course, a student must know:

- theoretical foundations of the finite element method in geotechnical calculations;
- approaches to modeling soil conditions and structural elements of structures in a three-dimensional environment;
- types of soil models (Mohr–Coulomb, Hardening Soil, Soft Soil, HS Small, etc.) and the conditions of their application;
- requirements of national and European regulatory documents for geotechnical design.

Competencies

As a result of the course, the student will acquire:

- **professional competencies** – ability to perform engineering analysis of complex geotechnical systems, skills to integrate the results of numerical modeling into design and operation of structures;
- **research competencies** – ability to perform sensitivity analysis of models, justify the selection of soil parameters, and critically evaluate the adequacy of the obtained results;
- **regulatory competencies** – ability to apply national (DBN, DSTU) and European (Eurocode 7) standards correctly when performing geotechnical calculations;
- **organizational and digital competencies** – ability to work with engineering software packages, integrate them into the BIM environment, and participate in team projects.

2. Description of the problem

In modern construction, one of the most complex tasks is to ensure reliable interaction within the ‘base–foundation–structure’ system. Incorrect assessment of soil properties, underestimation of spatial effects, or an oversimplified modeling approach may lead to excessive settlements, cracking of structural elements, or even emergency situations. Traditional two-dimensional analysis methods do not always allow consider of all factors, particularly under complex geological conditions, in case of multi-level excavations, underground structures, or hydraulic facilities. Meanwhile, the BIM approach is widely adopted in construction, requiring the integration of geotechnical data into a unified digital model that combines architectural and structural solutions.

PLAXIS 3D is a powerful tool for three-dimensional numerical modeling of soil masses and structures using the finite element method.

It makes possible to accurately simulate the behavior of the ‘base–foundation–structure’ system, analyse settlements and tilts, study the stability of excavations, and determine the stress–strain state of the foundation and the bearing capacity of foundations under various engineering and geological conditions. In BIM, the simulation results obtained in PLAXIS 3D can be integrated into the digital models of buildings and structures developed in Revit, Tekla, or other platforms, which contributes to the consistency and comprehensiveness of design solutions. It is essential to solve the problem of insufficient integration of the results of three-dimensional geotechnical modeling of the ‘base–foundation–structure’ system into BIM environment. As a result, geotechnical and structural aspects of design are considered separately and collaboration among different members of the project teams is not very efficient. Thus, it is necessary to:

- develop approaches to constructing geotechnical models in PLAXIS 3D that take into account the interaction of base, foundation, and structure;
- ensure the integration of the obtained results into BIM systems for comprehensive analysis;
- develop the skills of future engineers to use three-dimensional geotechnical modeling data in making structural and technological solutions;
- train students to apply Eurocode standards and national regulations in combination with BIM technologies to enhance the reliability and cost efficiency of projects.

3. Theoretical background

In the process of numerical modeling of geotechnical problems in PLAXIS software, one of the key stages is the selection of an appropriate soil model. The accuracy of calculation results and possibility to take into account actual behavior of the soil foundation under loading depend on it. The variety of models [3] available in PLAXIS (Mohr–Coulomb, Hardening Soil, Soft Soil, HS Small, etc.) complicates the selection of the optimal one for specific conditions.

The most widely used model for simulating soil deformation is the *ideal elasto–plastic Mohr–Coulomb model*, which serves as a basic model with a fixed yield surface fully defined by its parameters and independent of plastic deformation. For the elasto–plastic Mohr–Coulomb model, only four parameters are required, which can be determined through standard soil sample tests (strength and deformation characteristics): the modulus of deformation (E); Poisson’s ratio (ν); cohesion (c); and the internal friction angle (φ). This model describes the relationship between stresses and strains during the elastic and perfectly plastic stages of soil mass behavior, as illustrated by the curve in Fig. 1.1. Stress is directly proportional to strain until the limit stress is reached, after which the curve becomes completely horizontal.

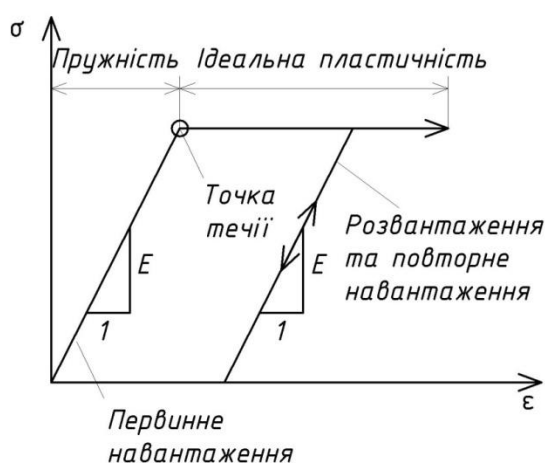


Fig. 1.1 Stress–strain curve of the Mohr–Coulomb model

The Mohr–Coulomb failure criterion is described by the following equation

$$|f| = f(\sigma)$$

A simpler form of the Mohr failure envelope is a straight line defined by the following condition:

$$|\tau| = \sigma (\operatorname{tg}\varphi) + c$$

Although the ideal elasto–plastic model is used in many modern geotechnical software applications, it has several significant drawbacks, including the use of the same value of the deformation modulus for initial loading, unloading, and reloading; the second principal stress σ_2 ($\sigma_1 \geq \sigma_2 \geq \sigma_3$) has no effect on material failure; and the meridians and the ultimate Mohr circle envelope are represented by straight lines.

Among nonlinear soil models used for numerical analysis, special attention should be given to a *nonlinear elasto–plastic model with soil hardening* (HS model), developed in 1999 and based on the theory of plastic hardening. It is an advanced model designed to simulate the behavior of various types of soils, both weak and strong. The behavior of the soil model during isotropic compression and shear is described by independent functions.

Plastic shear and volumetric strains are described independently by two functions: deviatoric stress function f_s (1.1) and isotropic stress function f^c (1.2). The Mohr–Coulomb criterion is used as the failure surface.

For deviatoric stress (Fig. 1.2), the yield function f^s is used:

$$f^s = \frac{1}{2E_{50}} \cdot \frac{q}{1 - q/q_a} - \frac{2q}{E_{ur}} - \gamma_p \quad 1.1$$

where E_{50} – secant modulus of deformation obtained from triaxial compression tests; E_{ur} – modulus of deformation during unloading and reloading; q – stress deviator; q_a – asymptote limiting the maximum deviatoric stress q_f .

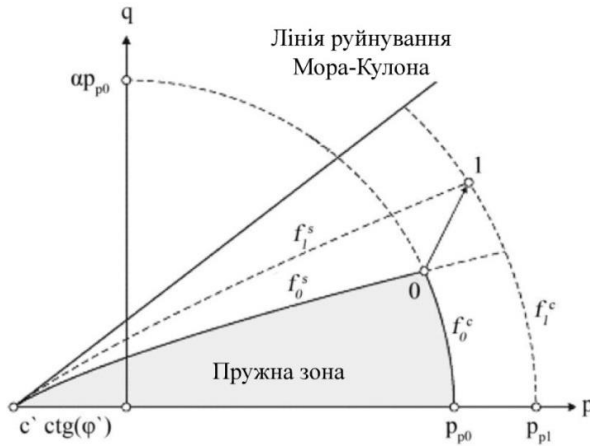


Fig. 1.2 Yield surfaces of the HS model for deviatoric (f^s) and isotropic (f^c) stresses

The hardening parameter γ_p is determined for the case of deviatoric loading according to the following formula:

$$\gamma_p = \varepsilon_1^p - \varepsilon_2^p - \varepsilon_3^p \approx 2 \cdot \varepsilon_1^p$$

In the case of deviatoric stress, dilatancy processes are represented within the framework of a non-associated flow rule, whereby the yield function and the plastic potential function do not coincide $g^s \neq f^s$.

The process of plastic volumetric deformation under isotropic loading is described by the yield function f^c (Fig. 1.2):

$$f^c = \frac{q^2}{\alpha^2} - p^2 - p_p^2 \quad 1.2$$

where, p_p - isotropic preconsolidation pressure, which depends on the soil formation. The stress–strain state is described by the Duncan–Chang hyperbolic model.

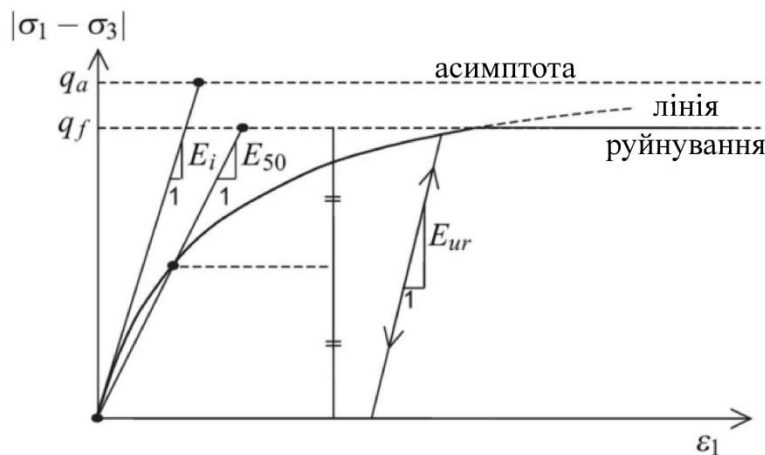


Fig. 1.3 Hyperbolic stress and strain relationship

The model employs different deformation moduli for the loading paths: a secant modulus of deformation obtained from triaxial compression tests for the initial loading path, and a separate modulus for the unloading and reloading paths.

The values of these deformation moduli increase with the acting minor principal stresses σ_3 (Fig. 1.3) and also depend on the strength characteristics of soils:

$$E_{50} = E_{50}^{ref} \left(\frac{c \cdot \cos \varphi - \sigma_3}{c \cdot \cos \varphi + p^{ref}} \right)^m ;$$

$$E_{ur} = E_{ur}^{ref} \left(\frac{c \cdot \cos \varphi - \sigma_3}{c \cdot \cos \varphi + p^{ref}} \right)^m ,$$

Oedometer modulus of deformation E_{oed} is determined by the following formula:

$$E_{oed} = E_{oed}^{ref} \left(\frac{c \cdot \cos \varphi - \sin \varphi \cdot \sigma_1}{c \cdot \cos \varphi + p^{ref} \sin \varphi} \right)^m ,$$

where m – a parameter indicating the degree of dependence of the deformation modulus on the acting stress (ranging from 0.45 to 1.0).

The advantages of the HS model include stress-dependent stiffness and consideration of the difference in stiffness between initial loading of soil and its subsequent unloading and reloading.

In HS model, the bulk and shear moduli are not constant but depend on the current stress level, which more accurately reflects the real behavior of the soil.

The course is based on students' background knowledge of the following disciplines: Physics, Theoretical Mechanics, Higher Mathematics, Strength of Materials, Structural Mechanics, Fundamentals of Soil Mechanics, Bases and Foundations, Engineering Geology, Engineering Surveys, and Fundamentals of Computer-Aided Design in Construction.

4. Situation analysis

4.1. Creation of a geometric model of the foundation and structural elements of a building

Launch the PLAXIS 3D program [7, 9, 10]. A *Quick Select* dialog window will appear on the screen, where you can either select an existing project or create a new one (Fig. 1.4).

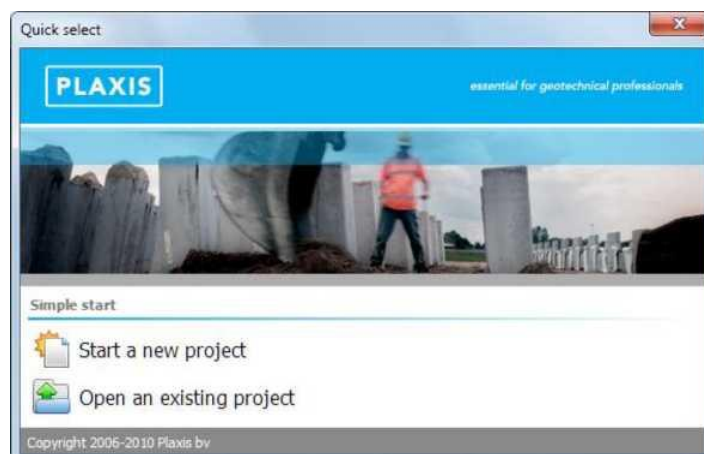


Fig. 1.4 *Quick Select*

Start working on a new project (*Start a new project*). The *Project Properties* window will open automatically, containing two tabs: *Project* and *Model*.

Project properties

The first step for all calculations is to enter the main parameters of the finite element model. The input of these primary parameters is carried out in the *Project Properties* window (Initial project data). These data include information about the project, units of measurement, and, the most important - the **dimensions of the soil model in plan view**. To enter the corresponding parameters for the calculation, the following steps must be performed:

In the *Project* tab, enter the project title (*Title*) as 'Foundation. Lesson 1' (using Latin characters only), and in the *Comments* field, type 'Settlement of a foundation' (also using Latin characters) (Fig. 1.5).

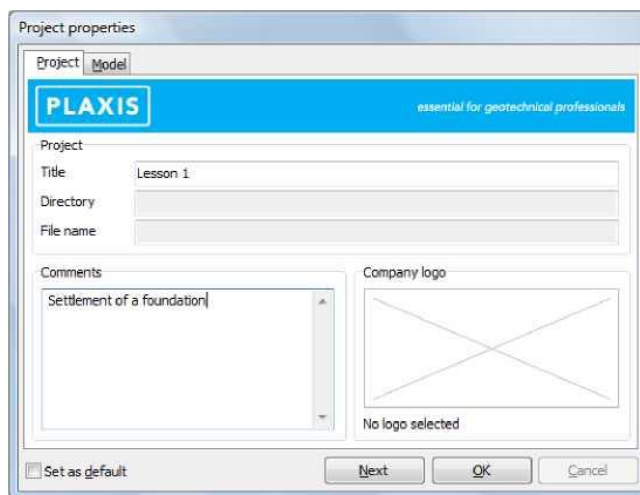


Fig. 1.5 *Project* tab of the initial project data

- ▶ Go to the *Model* tab (Fig. 1.6).
- ▶ In the *Units* field, keep the default standard units of measurement: *Length* measured in meters (m), *Force* in kilonewtons (kN), and *Time* in days (day).
- ▶ In the *General* group block (*General properties*), a fixed value of gravitational force directed downward along the vertical axis (-Z) is defined. In the *Earth gravity* field, the gravitational acceleration can be entered as 9.810 m/s². In the *y_{water}* field, the unit weight of water can be specified as $y_{\text{water}} = 10 \text{ kN/m}^3$.
- ▶ In the *Contour* group block, the boundaries of the soil model are specified as follows: $x_{\text{min}}=0$, $x_{\text{max}}=75$, $y_{\text{min}}=0$, $y_{\text{max}}=75$.
- ▶ To confirm the entered data, click the *OK* button.

Note! In case of an input error or if it is necessary to modify the project data, you can access the '*Project Properties*' window by selecting the corresponding option from the *File* menu.

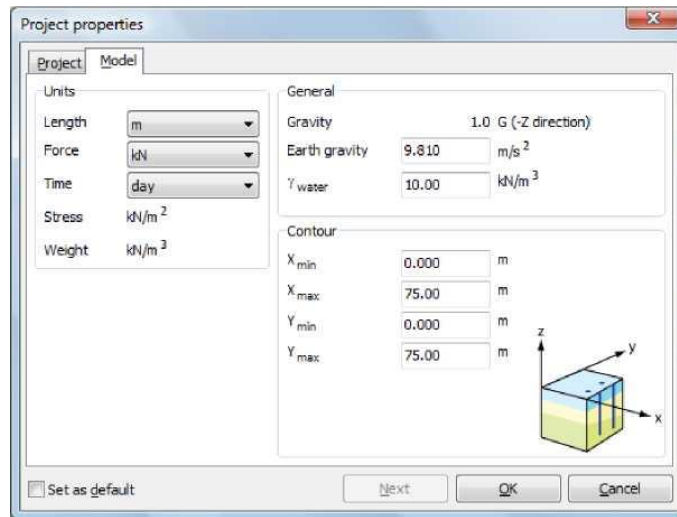


Fig. 1.6 *Model* tab of the initial project data


4.2. Building of an engineering geological section and input of the groundwater level

Information on the stratification of engineering geological layers by depth is entered through geological boreholes (hereinafter referred to as the geological log).

Geological logs represent points within the drawing area that contain information about the soil layering and groundwater level. If several geological logs are defined, PLAXIS 3D automatically interpolates the data between them to determine the soil stratification pattern at intermediate points.

Note! The PLAXIS 3D program can also process discontinuous layers, i.e., layers that are present only in certain geological logs.

In this example, there is only one soil layer, and therefore only one geological log is required to define the engineering geological section of the soil. To create the geological log, perform the following steps:

Click the *Create borehole*  button on the side toolbar to start defining the geological conditions. Place the cursor and click in the geometric model at the point with coordinates $(0, 0, 0)$. The geological log will be placed at the point with coordinates $(x, y) = (0, 0)$. A window titled *Modify soil layers* will appear on the screen.

In the *Modify soil layers* window, click the *Add* button to create a new soil layer. Leave the upper boundary *Top* unchanged ($z = 0$) and set the lower boundary (*Bottom*) to $z = -40$ m.

Set the groundwater level (*Head*) for the defined geological log to -2 m (Fig. 1.7). The creation of a set of engineering geological parameters and their assignment will be discussed in the next section.

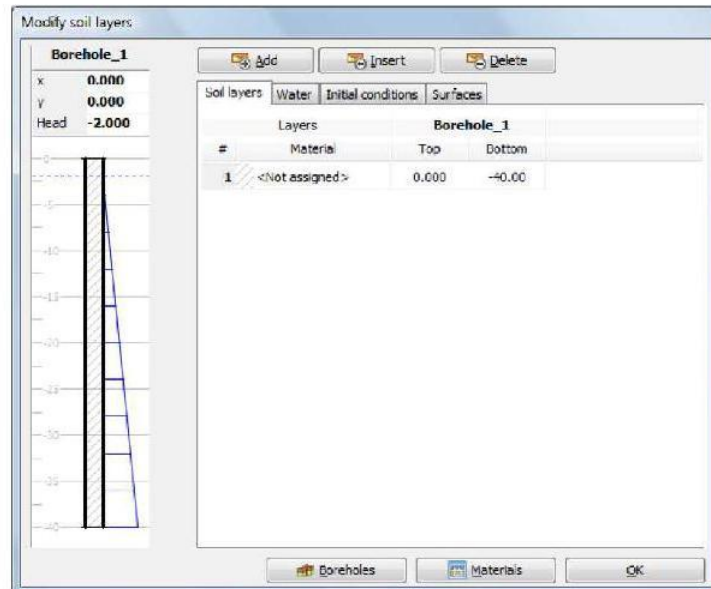


Fig. 1.7 *Modify soil layers*

4.3. Creation of a set of engineering geological element properties

To simulate the behavior of soil and structural elements, the corresponding material models and their parameters must be assigned to the elements of the geometric model to calculate. In PLAXIS, soil properties can be entered and stored in the program's database libraries according to the type of soil. The properties entered from the PLAXIS database can be assigned to one or more geometric elements (clusters). For different structural elements (beams, slabs, etc.), the same data set creation system is used; however, different types of structures have distinct parameters and, accordingly, different sections and input templates for defining their properties.

PLAXIS 3D distinguishes data sets by material type for soils and interfaces, plates, geogrids, beams, embedded piles, and anchors. Before generating the finite element mesh, it is necessary to assign the corresponding material properties to the materials.

Open the *Material Sets* window  by clicking the *Materials* button.

► Click the *New* button located at the bottom of the *Materials* window. A *Soil* window will appear on the screen, containing five tabs: *General*, *Parameters*, *Flow parameters*, *Interfaces [6]*, and *Initial*.

► In the *Identification* field of the *Material Sets* group block on the *General* tab, enter the soil name 'Clay' (Fig. 1.8).

► In the drop-down menu *Material model*, select the *Mohr–Coulomb* calculation model, and in the drop-down menu *Drainage type*, choose *Drained*.

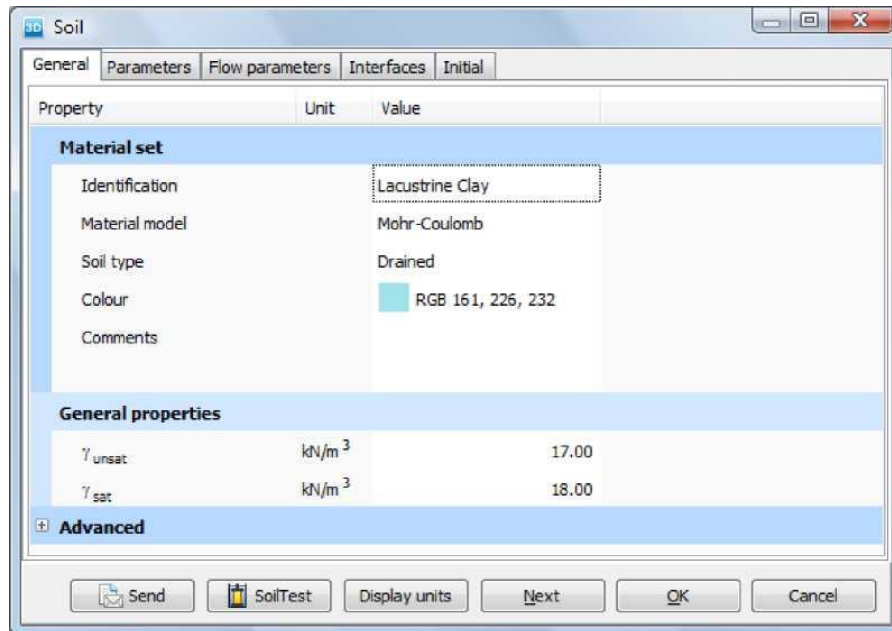


Fig. 1.8. General tab of Soils & Interfaces

► In the *General properties* group block, enter the values of the unit weight according to the material data set presented in Table 1.1. Leave the *Advanced* options set to their default values.

► Click the *Next* or *Parameters* button to continue entering the model parameters. The parameters displayed on the *Parameters* tab depend on the selected soil material model (in this case, the *Mohr–Coulomb* model). The *Mohr–Coulomb* model includes only five primary parameters (E' , ν' , c' , ϕ' , ψ'). A detailed description of various soil models and their corresponding parameters can be found in the Material Models Manual.

► Enter the model parameters E' , ν' , c'_{ref} , ϕ' i ψ for *Clay* according to Table 1.1 into the corresponding fields of the *Parameters* tab (Fig. 1.9).

► In this example, soil consolidation will not be considered. Therefore, soil permeability will not affect the obtained results, and the *Flow parameters* tab can be skipped.

► Since the geometric model does not contain interfaces (imaginary slip surfaces), the *Interfaces* tab can also be skipped.

Table 1.1

Soil and concrete material properties of the building

Parameters	Designation	Clay	Building	Unit of measurement
General properties				
Material model	<i>Model</i>	<i>Mohr–Coulomb</i> model	Linear-elastic model	-
Material behaviour type	<i>Type</i>	Drained	Non-porous	-
Unit weight of soil above the groundwater level	γ_{unsat}	17,0	50	N/m^3 ^κ
Unit weight of soil under the groundwater	γ_{sat}	18,0	-	N/m^3 ^κ

level				
<i>Parameters</i>				
Young's modulus (constant value)	e'	$1 \cdot 10^4$	$3 \cdot 10^7$	N/m^2 ^K
Poisson's ratio	ν'	0,3	0,15	-
Cohesion (constant)	c_{ref}	10	-	N/m^2 ^K
Friction angle	φ'	30,0	-	-
Dilatancy angle	ψ	0,0	-	-
<i>Initial</i>				
K_0 determination method		<i>Automatic</i>	<i>Automatic</i>	
Coefficient of lateral earth pressure	K_0	0,5000	1,000	-

► Click the *Initial* button and verify that automatic assignment of K_0 (*Automatic* option) is enabled. In this case, K_0 is determined using Jaky's formula: $K_0 = 1 - \sin\varphi$.

► Click the 'OK' button to confirm the entry of the current material data set. The created data set will appear in the tree list of material sets in the *Material Sets* window.

Create the *Clay* data set in the *Material Sets* window and, while holding down the left mouse button, drag it onto the image of the geological log on the left side of the *Modify soil layers* window. Release the left mouse button.

Note that the cursor changes its shape depending on whether it is possible to assign the data set. When the data set is assigned correctly, the soil color changes accordingly.

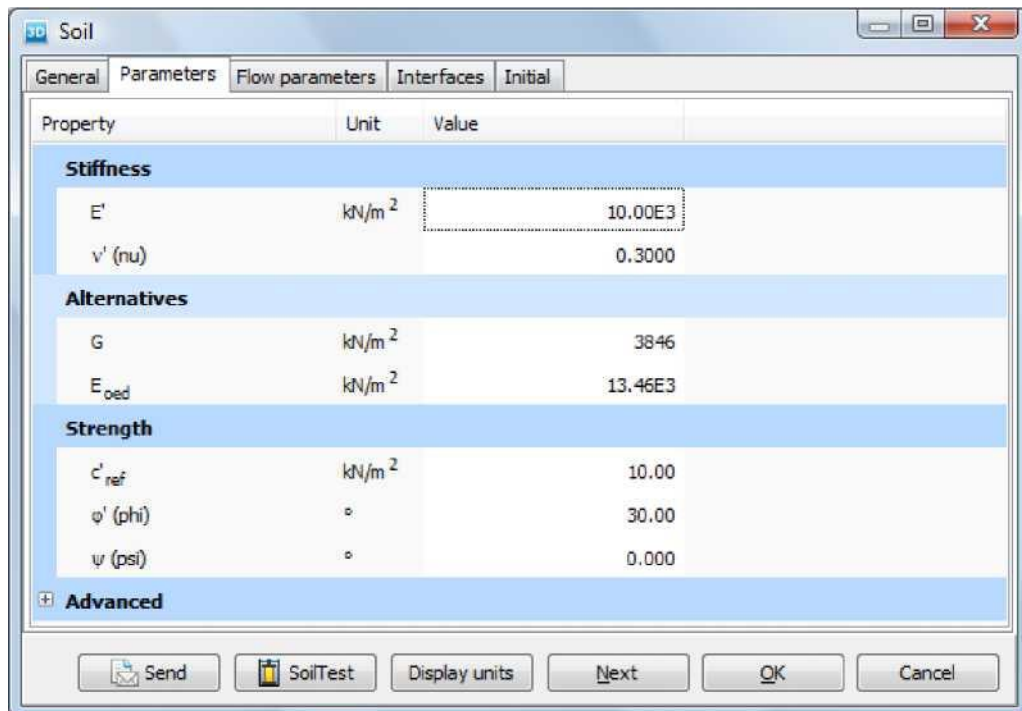


Fig. 1.9 Parameters tab of Soils & Interfaces

The building is modeled using a linear elastic (isotropic) material.

To create the corresponding data set, perform the following steps:

▶ Click *New* button in *Material sets window*.

▶ In the *Identification* field of the *Material Set* group block, *General* tab, enter the word '*Building*'.

▶ In the drop-down menu *Material model*, select the *Linear elastic* model, and in the *Drainage type* drop-down menu, choose the *non-porous* option.

▶ In the *General properties* group block, enter the unit weight according to the values presented in Table 1.1. The unit weight corresponds to the sum of the permanent and variable loads from the building.

▶ Click the *Next* button or the *Parameters* tab to continue entering the model parameters.

The linear elastic model has only two main parameters (E , ν).

▶ Enter the model parameters from Table 1.1 into the corresponding fields of the *Parameters* tab.

▶ Click the *OK* button to confirm the entry of the current material data set.

The created data set will appear in the *Material sets* window but will not be used directly.

▶ Click *OK* button to close the *Material sets window*.

▶ Click the *OK* button to close the *Modify soil layers* window.

Note! *PLAXIS 3D* distinguishes between the *project* and *global material databases*. A data set from one project can be used in another.

To view the *global database* in the *Material sets* window, click the *Show global* button.

4.4. Definition of structural elements

Structural elements are created in the *Structures* mode of the *PLAXIS* program.

To start defining the structural elements, click the *Structures* button.

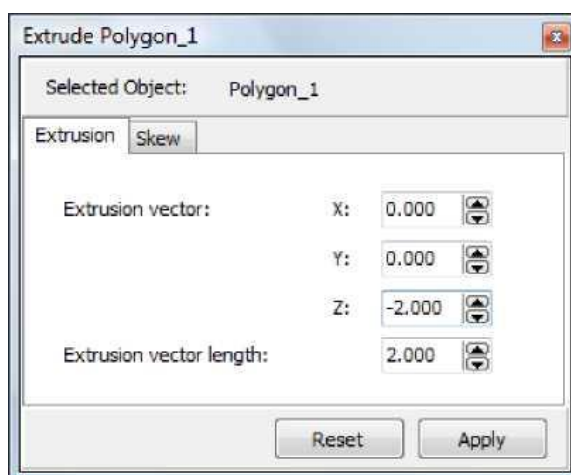



Fig. 1.10. *Extrude* tab

To model the building, click *Create surface*  and place the cursor at the point with coordinates $(0, 0, 0)$.

After clicking the left mouse button, the first point of the surface will be defined.

In the same way, define the other three points with the following coordinates, respectively: (0_{18_0}) , (18_{18_0}) , (18_0_0) . Click the right mouse button or the *Esc* key to finish defining the surface. Note that the created surface remains *selected* and is displayed in *red*.

▶ Click the *Extrude object* button (Видавити об'єкт) to create a solid volume from the surface.

► In the *Extrude* window (see Fig. 1.10), change the z value to - 2.

To close the window, click the *Apply* button.

► Click the *Select* button. Right-click on the created surface.

From the menu, choose the *Delete* option. This will delete the surface, but the building volume will remain.

The input of structural elements and the overall model construction have been completed. The next step is to generate the finite element mesh.

4.5. Generation of the Finite Element Mesh

To switch to the *Mesh* mode, click the *Mesh* button. PLAXIS 3D performs the mesh construction and generation completely automatically. During this process, the geometrical model is divided into additional structural elements. During mesh generation, the condition of all geometric objects in the model is taken into account, ensuring that the finite element mesh accurately reflects the positions of layers (without averaging), loads, and structures. Local mesh refinement will be applied within the building.

To generate the mesh, it is necessary to take the following steps:

► Click the *Refine mesh* button on the side toolbar .

The building volume will be highlighted in *green*.

► Click the *Generate mesh* button on the side toolbar or select the *Generate mesh* option from the *Mesh* menu. For the *Element distribution* option, choose *Coarse* mesh division and click *OK* to start the mesh.

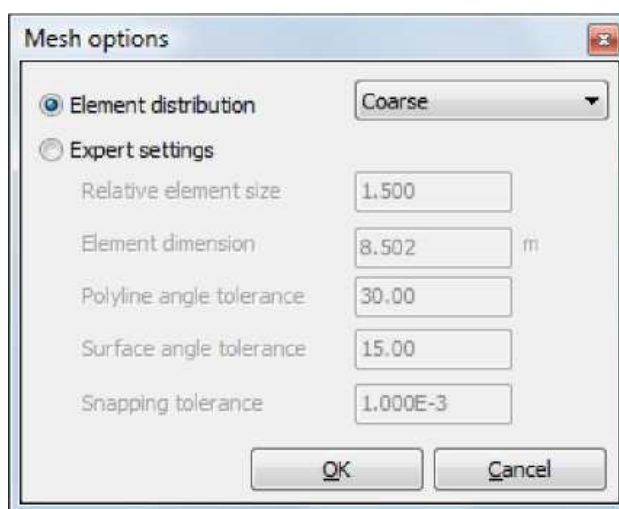



Fig. 1.11 *Mesh options*

After the mesh has been generated, click the *View* button. , the generated mesh will be displayed (see Fig. 1.12). To return to the *Mesh* mode, click the *Close* button.

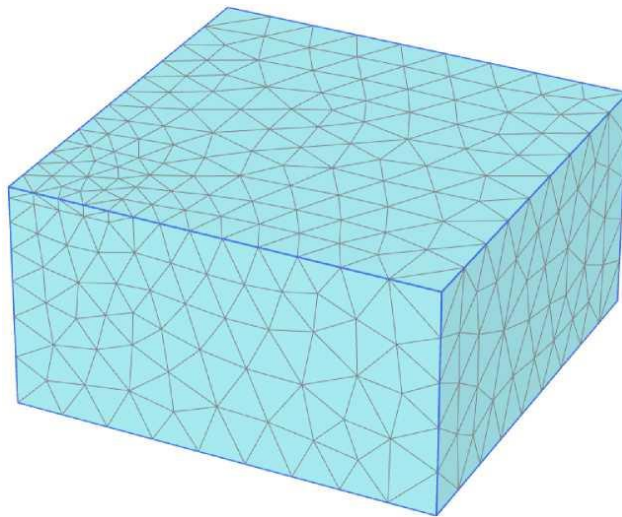


Fig. 1.12. Generated mesh in the Output window

4.6. Performing Calculations (Numerical Analysis of the Model)

The mesh generation completes the creation of the finite element model. Next, proceed to define the calculation phases by selecting the *Staged construction* option.

Initial conditions

In the *Initial phase*, the initial conditions are always defined. In general, the initial conditions include the geometrical configuration and the initial stress state, i.e., the natural stresses due to self-weight, pore pressure, and soil state parameters. The initial groundwater table (GWT) has already been defined in the *Modify soil layers* window. This level is taken into account when calculating the initial effective stress state of the soil. Therefore, there is no need to switch to the *Water levels* tab.

After a new project is created, the first calculation phase named '*Initial phase*' is automatically generated which is selected in the *Phases explorer* window (Fig. 1.13). All structural elements and loads present in the geometrical model are initially deactivated. Only the soil body is active at the beginning.

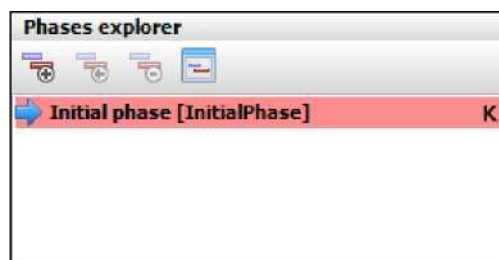


Fig. 1.13. *Phases explorer*

In PLAXIS 3D, two methods are provided for defining the initial stresses — the gravity loading method and the K_0 procedure.

By default, the K_0 procedure is selected for the calculation in the initial phase. Note that in the *Phases explorer*, this procedure is indicated by the letter 'K'.

Note! *The K_0 procedure can only be used for geometrical models with horizontal layer stratification, a horizontal base surface, and a horizontal groundwater table.*

To display the *Phases* window (Fig. 1.13), click the *Edit phase* button or double-click the desired phase in the *Phases explorer*.

- ▶ Click the *OK* button to close the *Phases* window.
- ▶ Check that all soil volume elements in the project are active and that the corresponding material *Clay* has been assigned to them.

Construction stage

After defining the initial conditions, the building can be modeled.

This will be done in a separate calculation phase, which should be added as follows:

- ▶ Click *Add* button in the *Phases explorer* window. A new phase titled *Phase_1* will appear.
- ▶ Double-click the desired phase to open the *Phases* window.

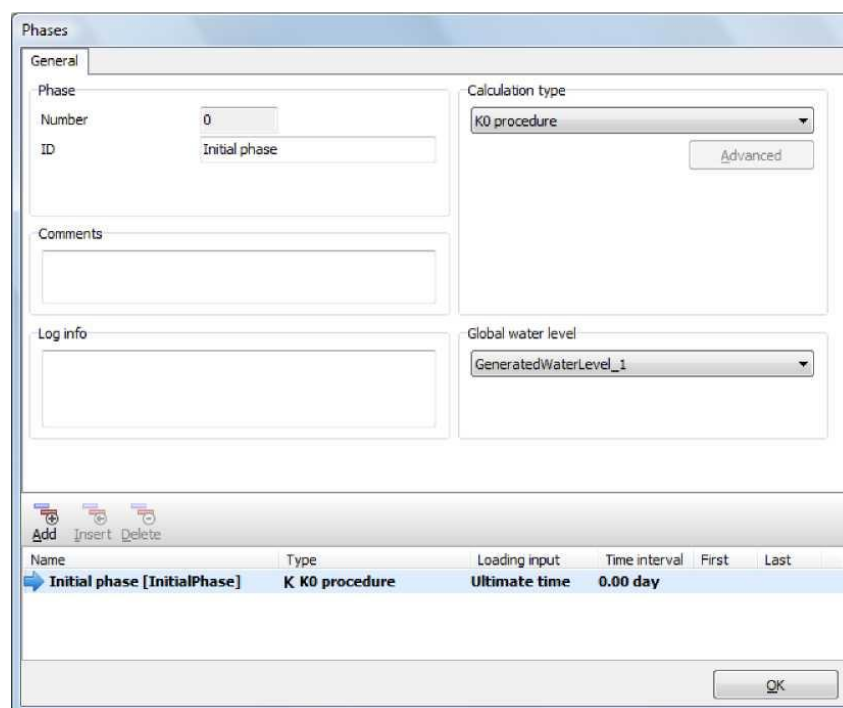


Fig. 1.14. General tab in the *Phases* window for the *Initial phase* calculation

- ▶ In the *General* tab, you can enter an appropriate name for the new phase in the ID field (for example, 'Building'). This field cannot be modified in the *Initial phase*.
- ▶ In the *Calculation type* field, select *Plastic*, and then click the *Parameters* tab to proceed to parameter definition.
- ▶ The *Parameters* tab (Fig. 1.15) contains the calculation control parameters. Keep the default settings in the *Iterative procedure* group block, and set the value in the *Additional steps* field to 250.
- ▶ The calculation parameters for the *Building* phase have been defined. Click the *OK* button to close the *Phases* window.
- ▶ Click on the right mouse button on the building volume. From the menu, select *Set material*, and then choose the *Building* option.

Performing calculations

All calculation phases (in our case, two phases) to execute are marked with a blue arrow. The execution order is controlled by the *Start from phase* parameter.

To start the calculation process, click the *Calculate* button.

Ignore the warning indicating that no nodes or stress points have been selected for curve plotting.

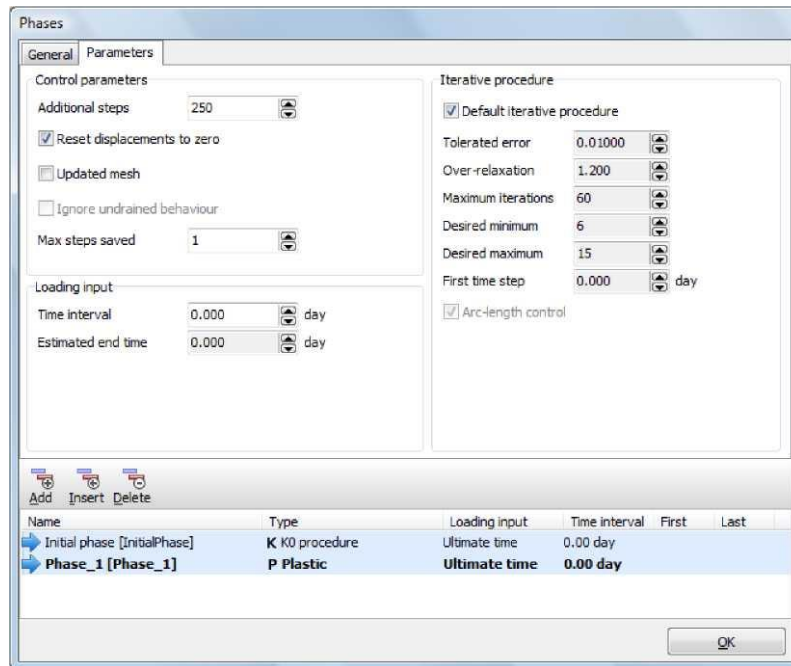


Fig. 1.15. *Parameters* tab in the *Phases* window for the *Building* calculation

During the calculation, a window appears on the screen to display information about the progress of the current calculation phase (see Fig. 1.16).

The continuously updated information in the *Active tasks* window contains data on the progress of the calculation process, such as the current calculation step number, the error of the current iteration, and other parameters.

The calculation process takes a few seconds. After the calculation is completed, the window will close, and the main window will become active again.

The program updates the list of phases in the *Phases explorer*, indicating with a *green check mark* that the calculation was completed successfully. A *red cross* marks a phase whose calculation failed to complete successfully.

Before viewing the obtained results, save the project.

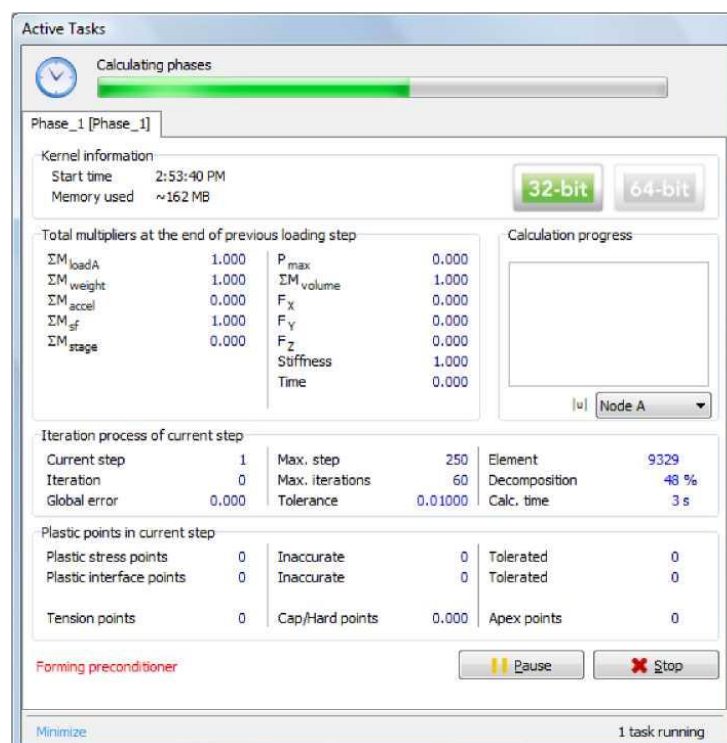


Fig. 1.16. *Active tasks* window displaying the calculation progress

Viewing calculation results

After the analysis (calculation) is completed, the obtained results can be viewed using the Output program. This program allows you to examine displacements and stresses throughout the entire three-dimensional model, as well as in individual cross-sections or structural elements. The calculation results can also be displayed in a table.

To view the current results, it is necessary to:

- ▶ select the last calculation phase (Building) in the Phases explorer.
- ▶ Click the *View calculation results* button on the side toolbar to open the Output program.
- ▶ From the *Deformations* menu, select the *Total Displacements option*. The total displacements will now be displayed in different colors (Fig. 1.17).

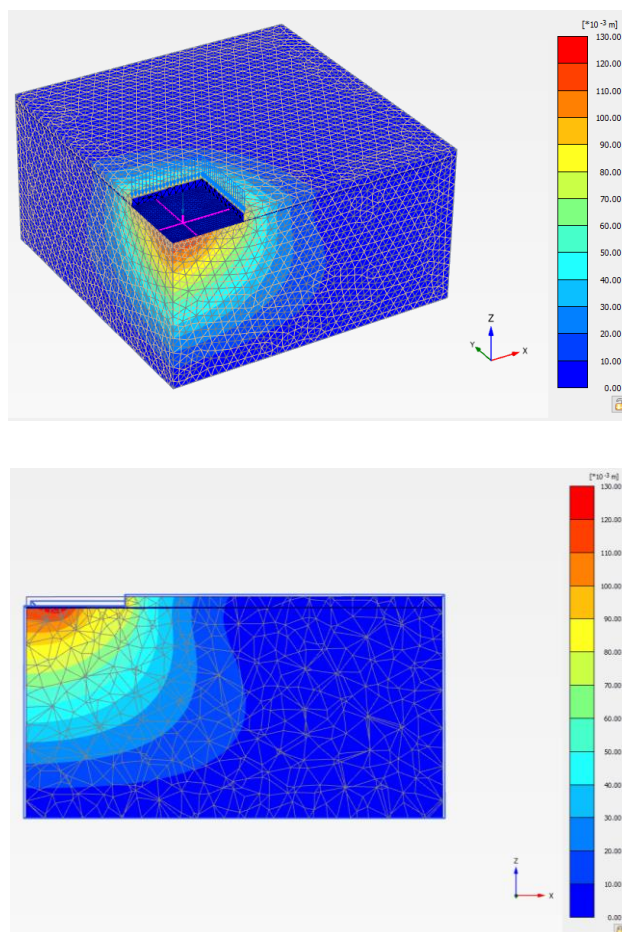


Fig. 1.17. Distribution pattern of total displacements for the final calculation phase

The displacement values within the model are indicated on the legend scale by corresponding colors. If the legend scale is not visible, select the *Legend* option from the *View* menu.

In the *Output* window, click the *Iso-surfaces* button to display the regions with equal displacement values.

5. Discussion questions

1. What is the main difference between PLAXIS 2D and PLAXIS 3D?
2. What types of problems are the most appropriate to model using PLAXIS 3D?
3. Why is it important to consider three-dimensional nature of soil processes in geotechnical calculations?
4. Which input data are the most critical for calculations accuracy in PLAXIS 3D?
5. How should boundary conditions be properly defined for a model?

6. What are the potential risks of idealizing the soil as a linear-elastic model?
7. When is it advisable to apply advanced soil models (e.g., Hardening Soil, Soft Soil Creep, etc.)?
8. How can a pile group be modeled in PLAXIS 3D, and how can the interaction between piles be evaluated?
9. What are the key aspects of modeling foundation behavior on weak soils in a three-dimensional model?
10. How can BIM data (e.g., from Revit or Civil 3D) be used in PLAXIS 3D to build a geotechnical model?
11. In what cases can a 3D model produce results that differ significantly from a 2D model?
12. How should the soil model dimensions be defined correctly in PLAXIS 3D?

Critical / reflective questions

13. Can the results of numerical modeling in PLAXIS 3D be considered the ‘ultimate truth’?
14. How can the results of field experimental tests (e.g., static pile load tests) be correlated with PLAXIS 3D models?
15. How can soil parameters be adapted or calibrated for use in PLAXIS 3D calculations?
16. Why do geotechnical systems require a specific approach to BIM modeling?
17. What limitations of the PLAXIS 3D software package do you consider the most significant?
18. How effective is the use of PLAXIS 3D in the educational process for training future civil engineers?

6. Possible solution strategies

Integration of PLAXIS 3D into educational programmes:

- including specialised courses and practical workshops on numerical modelling of geotechnical processes;
- using PLAXIS 3D as a core tool for course and diploma projects on foundations, excavation retaining structures, etc.;
- engaging students in the analysis of real engineering cases within a 3D environment.

Development of methodological materials:

- creating step-by-step guidelines for building a geotechnical model (geometry, mesh, boundary conditions);
- preparing video tutorials and training examples (foundation, excavation pit, pile groups, tunnel);
- creating libraries of basic soil parameter in accordance with DBN/EN standards and practical recommendations.

Practical integration of analysis (calculations):

- comparing PLAXIS 3D modelling results with field test data (e.g., static pile load test);
- comparing numerical results with hand-calculation methods for settlement, bearing capacity, and slope stability;
- conducting sensitivity analysis of soil parameters and boundary conditions.

Organisation of teamwork:

- training students in the collaborative use of PLAXIS 3D within interdisciplinary teams (geologists, structural engineers, design engineers);
- integrating PLAXIS results into BIM for coordination with architectural and structural models;
- establishing the practice of maintaining a change log and version control for a geotechnical model.

Research:

- investigation of the efficiency of different constitutive soil models (Mohr–Coulomb, Hardening Soil, Soft Soil Creep);

- validation of new soil parameter calibration methods based on field test results;
- analysis of the influence of spatial soil heterogeneity on calculation results in 3D models.

Digitalisation of operation:

- using PLAXIS 3D models to forecast long-term settlements and assess the condition of structures;
- training students to integrate calculation outputs with deformation and stress monitoring systems;
- developing skills in digital support of structures during operation and reconstruction.

7. Solution (example)

Using PLAXIS 3D to model geotechnical processes in the design of bases and foundations for case projects (e.g., industrial buildings or engineering structures) enables students to carry out a full cycle of numerical analysis of subsoil behaviour. Combining PLAXIS 3D with such software as Tekla Structures or LIRA-FEM ensures verification of the consistency of ‘base–foundation–structure’ interaction and the validity of computational models.

The use of PLAXIS 3D develops students’ competences to build three-dimensional geotechnical models, analyse stress–strain state of soils, and predict structural stability. This approach fosters engineering thinking and understanding the physics of soil processes, enabling not only to model but also critically evaluate the obtained results. It meets modern requirements for training professionals in geotechnical engineering and digital design.

The practical integration of calculations into educational process involves comparing modelling results with field test data, correlating numerical outcomes with analytical methods, and conducting sensitivity analysis of soil parameters.

It is also important to organise students’ teamwork in interdisciplinary projects that combine geotechnical modelling in PLAXIS with BIM, including the practice of version control and maintaining a change log. Research should focus on comparing validated soil models, calibrating parameters based on field data, and analysing spatial heterogeneity. In the process of digitalisation of the operation and maintenance (O&M) of structures, PLAXIS 3D can be used to forecast long-term settlements, integrate with deformation monitoring systems, and develop skills in digital lifecycle support of structures.

8. Practical assignment for students

To complete the assignment, you must select the initial data for your individual variant from Tables 3.1–3.4 and carry out the assigned task (Section 4).

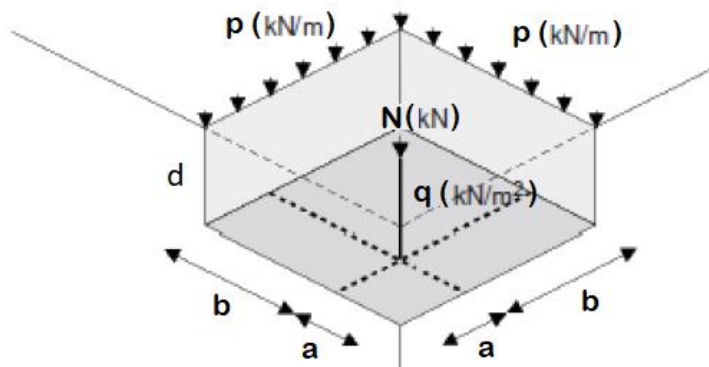


Fig. 2.1. Geometric model of the basement

Table 8.1

Initial data for the assignment

№ variant	Slab size, m×m	a, m	b, m	N, κN	q, κN/m ²	p, κN/m	d, m
1	17×17	5	1 2	1 1000	5,3	360	2
2	16×16	4	1 2	1 0000	5,1	370	2
3	15×15	3	1 2	9 000	4,9	350	2
4	14×14	2	1 2	8 000	5,4	310	1
5	13×13	5	8	7 000	5,2	240	1
6	12×12	4	8	6 000	5,3	250	1
7	11×11	5	6	9 000	5,0	300	2
8	10×10	5	5	8 000	4,8	320	2
9	19×19	6	1 3	7 000	4,5	280	1
10	20×20	6	1 4	6 000	4,7	200	1

In the drop-down menu *Material model*, select the *Mohr–Coulomb* computational model, and in the drop-down menu *Drainage type*, choose *Drained*.

Table 8.2

Physical and mechanical properties of the soil

Silty clay soil	Soil description	Variant									
	Dimensional quantity	№1	№2	№3	№4	5	6	7	8	9	10
Unit weight of soil above the groundwater table	γ_{unsat} , κN/m ³	17,2	16,9	16,7	17,1	7,5	7,4	7,8	7,6	6,8	6,9
Unit weight of soil under the groundwater level	γ_{sat} , κN/m ³	18,2	18,9	17,7	18,1	8,5	8,4	8,8	8,6	8,8	8,9
Young's modulus	e' , κN/m ³	10000	11000	12000	9800	500	000	200	2500	0000	1500
Poisson's ratio	ν'	0,3	0,3	0,3	0,3	,3	,3	,3	,3	,3	,3
Cohesion	c_{ref} , κN/m ³	10	15	14	12	1	6	3	4	2	1
Friction angle	φ'	30,0	25	23	26	7	8	9	2	2	2

Dilatancy angle	ψ	0,0	0,0	0,0	0,0	0,0	0,0	0,0	0,0	0,0	0,0
K_0 determination method		<i>Automatic</i>									
Coefficient of lateral earth pressure	K_0	,50	,55	,60	,65	,59	,52	,58	,62	,61	,63
Material behaviour type	<i>Type</i>	Drained									

Table 8.3
Properties of basement floor slab and wall materials

Parameter	Designation	Basement floor	Basement wall	Unit of measurement
Thickness	d	0,5	0,3	m
Weight	γ	15	15,5	$\kappa N/m^3$
Behaviour type	<i>Type</i>	Linear,	Linear,	-
Young's modulus	E	$3 \cdot 10^7$	$3 \cdot 10^7$	$\kappa N/m^2$
Poisson's ratio	ν	0,15	0,15	-
Shear modulus	G	$1,304 \cdot 10^7$	$1,304 \cdot 10^7$	$\kappa N/m^2$

Table 8.4
Characteristics of basement column and beam materials

Parameter	Designation	Basement column	Basement beam	Unit of measurement.
Cross-sectional area	A	0.49	0.7	m^2
Bulk density	γ	24.0	6.0	$\kappa N/m^3$
Behaviour type	<i>Type</i>	linear	linear	-
Young's modulus	E	$3 \cdot 10^7$	$3 \cdot 10^7$	$\kappa N/m^2$
Moment of inertia	I_3	0.020	0.058	m^4
	I_2	0.020	0.029	m^4

Guidelines for the assignment (PLAXIS 3D):

1. Build a 3D geotechnical model in PLAXIS 3D.
2. Perform a static analysis (calculation) of the stress–strain state of soil foundation under the structural load.
3. Justify the choice of the soil model (Hardening Soil, Mohr–Coulomb, or another) and the parameters used for the analysis (calculation).
4. Prepare the results presented as visualisations: settlement contour plots; stress distribution within the soil mass; settlement graphs at control points.
5. Prepare a report and drawings based on the work completed.

9. Discussion and reflection

PLAXIS 3D software package for geotechnical modelling enables students not only to master modern tools of numerical analysis but also to rethink critically the approach to design of bases and foundations in complex engineering and geological conditions. It is important to consider how the role of a geotechnical engineer changes in the digital environment, which professional competencies become essential for working with BIM-oriented models, and what challenges arise when combining national standards [1] with Eurocode requirements in practical applications.

Questions to discuss:

1. What role does PLAXIS play in BIM-oriented design?
2. How does a BIM model the building differ from a geotechnical BIM model (soil, foundation, underground structures)?
3. How can the geometry of a foundation or pile group be imported from Revit or Civil 3D into PLAXIS 3D?
4. What data formats can serve as intermediate ‘bridges’ between BIM and PLAXIS?
5. How can modelling results (settlements, stresses, deformation curves) be visualised in the BIM environment?
6. In which cases the integration of BIM + PLAXIS is the most appropriate - excavations, underground structures, or high-rise buildings?
7. Can a BIM model ‘respond’ to changes in soil parameters if they are refined in PLAXIS?
8. How can PLAXIS results help structural engineers make more accurate decisions in the BIM environment?
9. What issues arise due to the lack of a unified standard for geotechnical data in BIM?
10. Is it possible to create a library of ‘geotechnical objects’ (soil layers, piles, anchors) compatible with both BIM and PLAXIS?
11. Can BIM without PLAXIS be considered an ‘incomplete digital model’ of a structure?
12. What are the main barriers to integrating PLAXIS into BIM - technical, economic, or methodological?
13. Should future engineers study both BIM and PLAXIS simultaneously to master a full cycle of digital design?
14. How realistic is it to create a unified digital twin of a structure that integrates architecture, structural engineering, and geotechnics?

10. Conclusions

The application of PLAXIS 3D in the educational process allow students to gain hands-on experience with modern methods of geotechnical modelling, including development of three-dimensional models of bases and foundations, analysis of the stress–strain state of soils and structures, as well as verification of computational schemes in accordance with DBN and Eurocode standards.

The case studies carried out in this software have demonstrated the importance of integrating BIM approaches into geotechnical studies, contributed to the development of skills in critical analysis of calculation results, and combined theoretical training with real needs in modern construction.

References

1. DBN V.2.1-10:2018. Bases and foundations of buildings. General provisions – Kyiv: Ministry for Regional Development, Construction and Housing and Communal Services of Ukraine, 2018. – 36 p.

2. Vynnykov Yu. L. Mathematical modelling of the interaction between foundations and compacted bases during construction and subsequent operation: Monograph. – Poltava: PoltNTU, 2016. – 280 p.

3. Brinkgreve R.B.J. PLAXIS 2D-version 9. Finite element code for soil and rock analyses / R.B.J. Brinkgreve, W. Broere, D. Waterman / User Manual – Rotterdam: Balkema – 2008.

4. Matviikiv O., Tkachenko S., Khakhanov V. Engineering design of complex objects and systems: Textbook. – 2016. – 261 p.

5. S. Yu. Saienko, I. V. Nechyporenko. Fundamentals of CAD: Textbook. – Kharkiv: KhDUKht, 2017. – 120 p.

6. PLAXIS 3D 2024.3 / General Information Manual. April. 03, 2025 – 34 p.

7. PLAXIS 3D 2024.3 / Tutorial Manual. April. 03, 2025 – 188 p.

Recommended for individual learning

8. Piling Engineering / K. Fleming. A. Weltman, M. Randolph, K. Elson. – London; New York: Taylor and Francis, 2008. – 398 p.

9. Guidelines for performing practical tasks using the PLAXIS 3D software package for the course ‘Modelling of foundation–soil interaction’ for Master’s degree full-time and part-time students of speciality 192 ‘Construction and Civil Engineering’/ Authors: Siedin V. L., Bikus K. M., Kovba V. V., Zahilskyi V. A., Kononov D. V. – Dnipro: SHEI PSACEA, 2019. – 31 p.

10. Guidelines for completing the assignments (test paper) for the course ‘Modelling of foundation–soil interaction’ for Master’s degree part-time students of speciality 192 ‘Construction and Civil Engineering’ / Authors: Siedin V. L., Bikus K. M., Kovba V. V., Zahilskyi V. A., Ulianov V. Yu. – Dnipro: SHEI PSACEA, 2022. – 28 p.

10. APPLICATION OF BIM IN STRUCTURAL DESIGN: CASE STUDY ON INTEROPERABILITY ANALYSIS, REINFORCEMENT DETAILING, AND QUANTITY TAKE-OFF

Tytarenko R. Yu., Chmil R. Ye., Pozniak O. R.

1. ABSTRACT

Building Information Modelling (BIM) is increasingly used in construction, offering a centralized virtual model with multi-disciplinary data. Full interoperability in structural design remains challenging due to current system limitations. A case study of a structural project evaluated the workflow from modelling and data transfer between Revit and Robot, through structural analysis and reinforcement detailing, to integrating results into the main BIM model. Limitations were partially overcome with automated transfer of drawings and material quantities, and additional BIM capabilities enabled the generation of specific drawings and reinforcement maps. The approach demonstrates an effective workflow that enhances efficiency and productivity in structural design offices.

2. INTRODUCTION

The Building Information Modelling (BIM) methodology is based on creating a centralized digital 3D model of a building or infrastructure, updated throughout the project lifecycle [2]. This model organizes data hierarchically, enabling retrieval by specialty, element, or material. BIM tools allow manipulation of information for activities such as graphical representation, material quantification, budgeting, and construction planning. Centralized information and collaborative platforms reduce errors and conflicts, saving costs across the building's lifecycle [4].

In structural design, parametric objects represent elements like slabs, beams, columns, and foundations, with properties such as elasticity, Poisson's ratio, and material density. Structural engineers generate the parametric model and transfer it to analysis software to validate the frame and produce drawings. Full interoperability is crucial, especially for transferring analysis results and reinforcement detailing back to the BIM model, which often represents the main challenge in BIM-based structural projects.

Current BIM tools lack efficient interoperability, leading to inconsistencies, omissions, and data errors during system model transfer. The main aim is to identify limitations at each step and propose strategies to reduce inaccuracies in the BIM process. While BIM offers clear benefits – such as improved collaboration, faster design changes, higher-quality parametric modelling, and better communication with clients and teams – its adoption requires technological investment, training, and organizational adjustments. Previous research at the University of Lisbon highlighted interoperability challenges between systems like Revit, Robot, SAP, ArchiCAD, and ETABS, particularly in transferring analytical models, loads, and reinforcement detailing. Despite these challenges, BIM enables structural engineers to efficiently analyse design alternatives, improve project accuracy, and reduce costly errors, though interoperability issues remain a key obstacle to broader adoption [5].

In the study by Sampaio and Gomes [3], the analysis focuses on interoperability between two integrated Autodesk programs: Revit, used for modelling, and Robot, used for structural calculations with direct access from Revit. Since both are from the same company, the data is transferred in its native format. A building case was used to explore modelling and analysis processes where data transfer occurs: a structural solution is modelled in Revit and the analytical frame is checked; the model is transposed to Robot and its geometric consistency verified; structural analysis and reinforcement detailing are performed in Robot; reinforcement data is transferred back

to Revit and checked; finally, the model is completed and drawings and maps are automatically generated [1].

At each stage, interoperability efficiency is assessed to determine which tasks are best performed in each software while ensuring data accuracy. Structural models, diagrams, reports, and 3D outputs are reviewed during transfers, and gaps, inconsistencies, and omissions are identified. Despite limitations linked mainly to interoperability, the process demonstrates clear advantages for developing structural designs on BIM-based platforms.

3. METHODS

3.1. Case Study Modelling

A single-family house in Algarve, Portugal, is the selected case for developing a structural project in a BIM environment (see Figs. 3.1, 3.2). The structural solution is digital drawings and a report detailing the calculation method and initial sizing. Drawings of the foundation and floors contain all necessary information to generate a representative BIM model.



Fig. 3.1. BIM Model Architecture

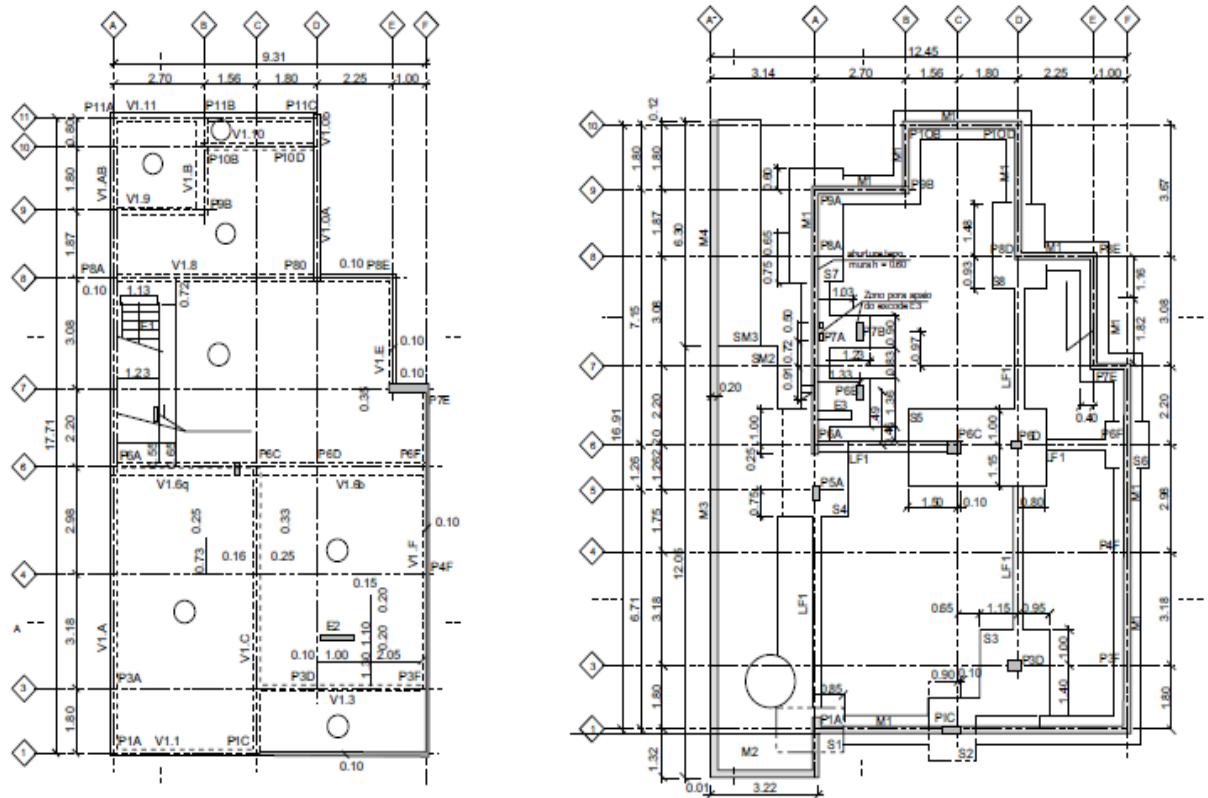


Fig. 3.2. Structural Design Technical Drawings

3.1.1. Structural Solution

The BIM model was created using Revit (Autodesk) based on the project's structural drawings. The villa has two above-ground floors and a basement, requiring four modelling levels: foundation, ground floor, first floor, and roof (see Figs. 3.3, 3.4). The structure is reinforced concrete (RC).

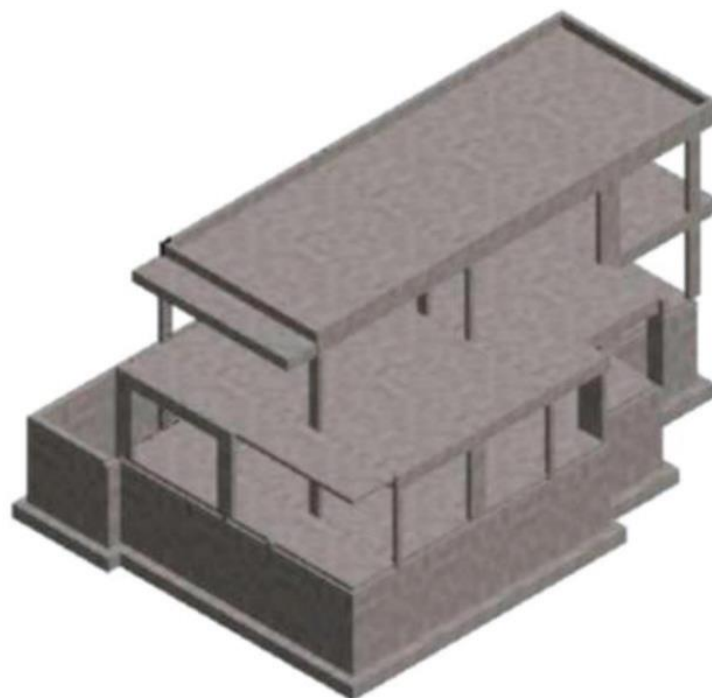


Fig. 3.3. 3D Perspective of the Structural BIM Model

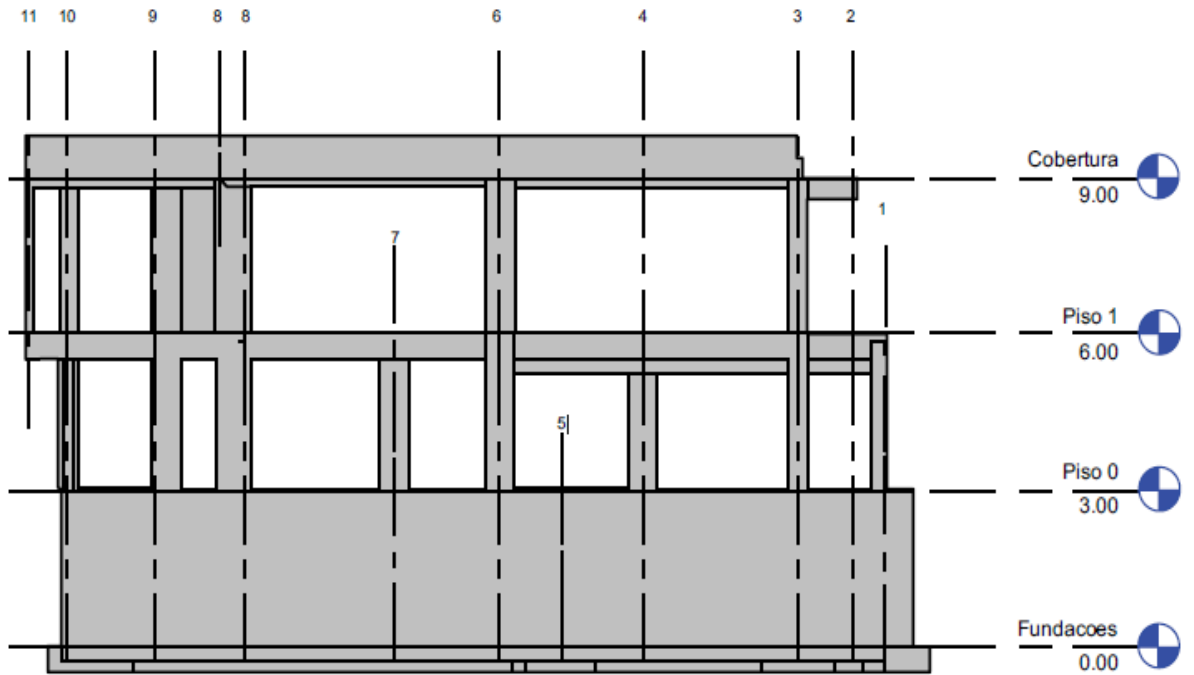


Fig. 3.4. Elevation View of the Structural BIM Model

During the model creation, parametric elements such as columns, load-bearing walls, beams, and slabs (see Figs. 3.5–3.7) were used, each linked to the corresponding material characteristics of concrete and steel (see Table 3.1).

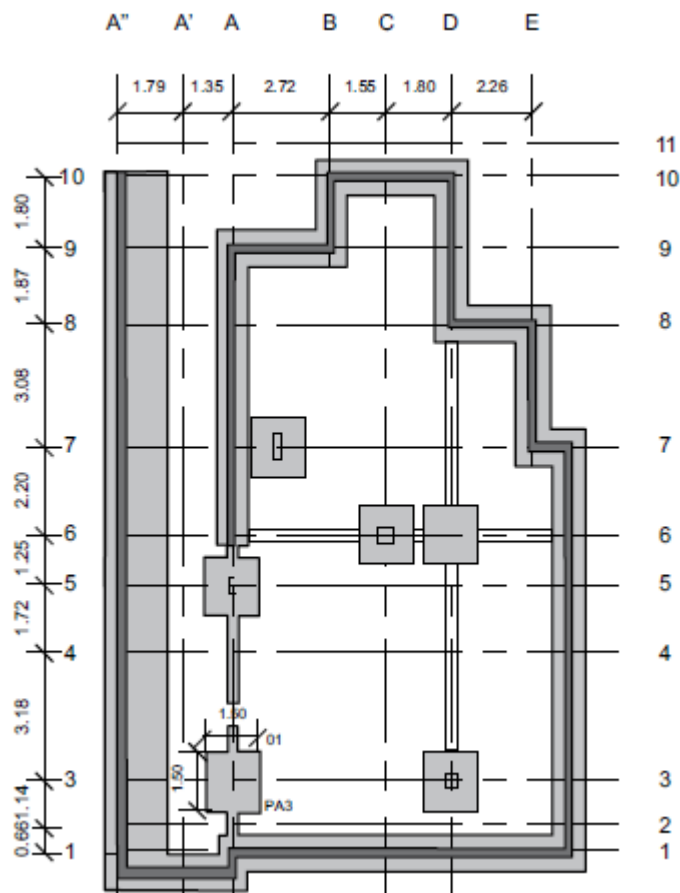


Fig. 3.5. Foundation Plan of the Structural Model

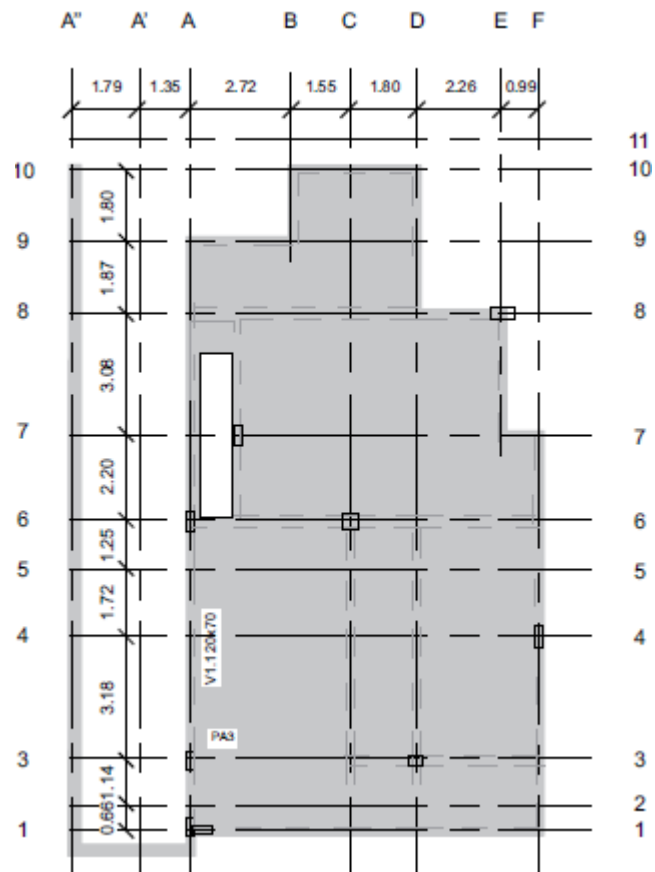


Fig. 3.6. 1st Floor Plan of the Structural Model

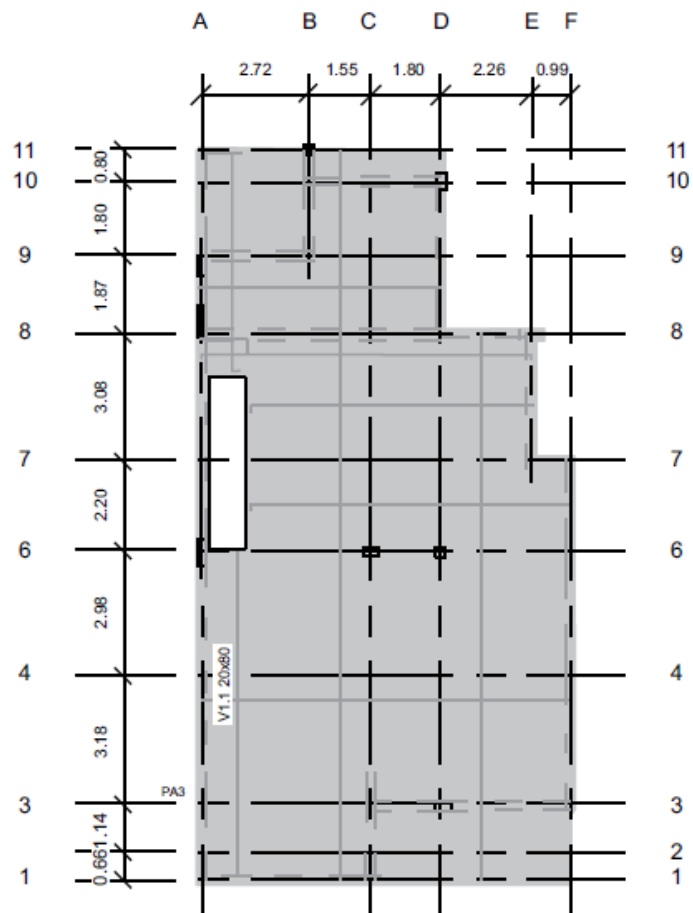


Fig. 3.7. 2nd Floor Plan of the Structural Model

Table 3.1.

Structural Material Characteristics

Material	f_{ck} MPa	f_{cd} MPa	f_{ctm} MPa	f_{cm} GPa	γ_c kN/m ³	f_{yk} MPa	f_{yd} MPa	E_s GPa	ϵ_y	γ_s kN/m ³
Concrete C30/37	30.0	20.0	2.9	3.0	24.0	—	—	—	—	—
Steel A500 NR SD	—	—	—	—	—	500	435	200	2.18×10^{-3}	30.0

The concrete slab is designed with beams and a uniform thickness of 160 mm across all panels on both floors. Peripheral load-bearing walls are provided on the first floor, while central columns, continuous through the upper levels, are included. The foundation system is simple, comprising strip footings beneath the peripheral walls and isolated footings under the central columns, all connected by foundation beams.

The model is generated using a grid of alignments placed directly on the imported DWG drawings (see Fig. 3.8). Modeling starts with columns, followed by walls, beams (see Fig. 3.9), slabs, and finally foundations (see Fig. 3.10).

The software provides parametric libraries of structural elements, each adjusted to the required section dimensions (see Table 3.2), thicknesses, and assigned material with proper mechanical properties.

Each structural family element contains the data needed for analytical discretization. A beam, for instance, is discretized along its axis (see Fig. 4.1) and linked to its cross-sectional area and inertia, key parameters for structural analysis.

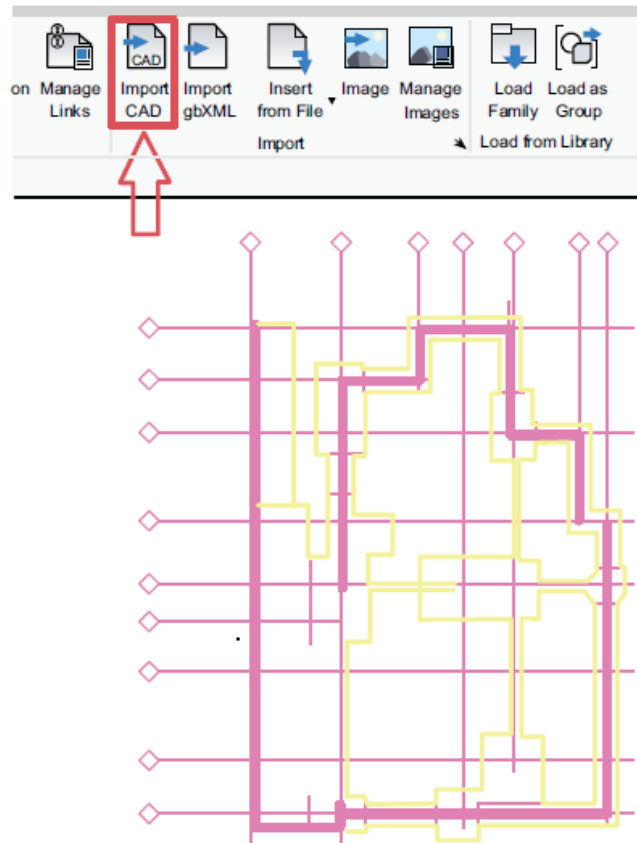


Fig. 3.8. Drawing Import Process in Revit

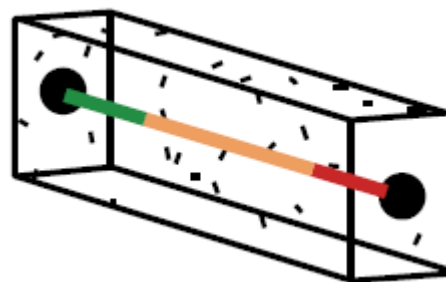


Fig. 3.9. Geometric and Analytical Representation of a Beam

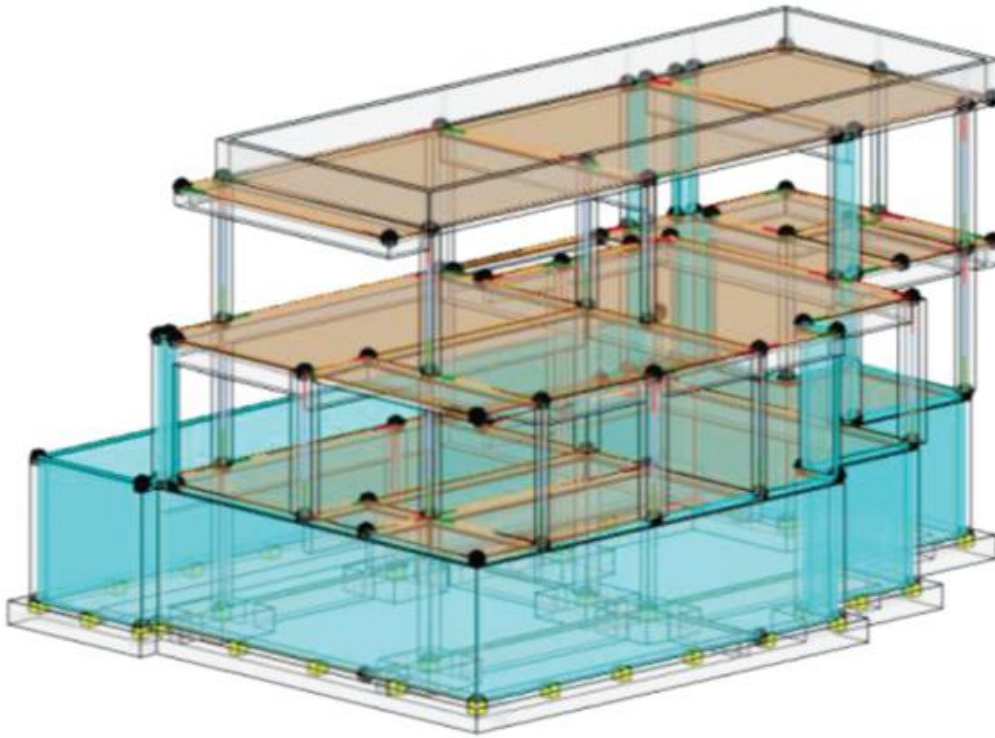


Fig. 3.10. Analytical Representation of the Structural Solution

Table 3.2.

Beam Creation in Revit Editor

Family	Concrete-Rectangular	Load
Type:	V20×30	Duplicate
		Rename
	Type Parameters	
	Structural	
	Section Shape	
	Dimensions	
	b	200
	h	500

In the modeling system, an initial geometric check is performed to verify the consistency of link nodes between elements, with corrections made based on the structure’s analytical representation. At any stage, the software can display the geometric and analytical models simultaneously and apply error-detection filters to identify inconsistencies in element connections, which the modeler must correct (see Fig. 3.11).

Loads can be applied directly in Revit, creating various permanent and variable loads and the necessary load combinations (see Table 3.3). However, only the Robot is used for seismic analysis, as it ensures the correct and complete assignment of loads and combinations according to current standards. Therefore, although Revit can represent most loads and combinations, all loads and combinations were applied in Robot (see Fig. 3.12).

Table 3.3.

Load Cases and Load Combinations Assigned to the Model

Load Cases				Category	
	Name	Case Number	Nature		
1	DL1	1	Dead	Dead Loads	
2	SC2	3	Live	Live Loads	
3	SC1	4	Live	Live Loads	
4	RCP	2	Dead	Dead Loads	
5	Modal	5	Dead	Dead Loads	
Load Combinations				Type	State
	Name	Formula			
1	ELU	$1.35 \times CL1 + 1.35 \times RCP + 1.5 \times RC2$		Comb.	Ultimate
2	ELS+CQ	$1.0 \times CL1 + 1.0 \times RCP + 0.3 \times RC2$		Comb.	Service

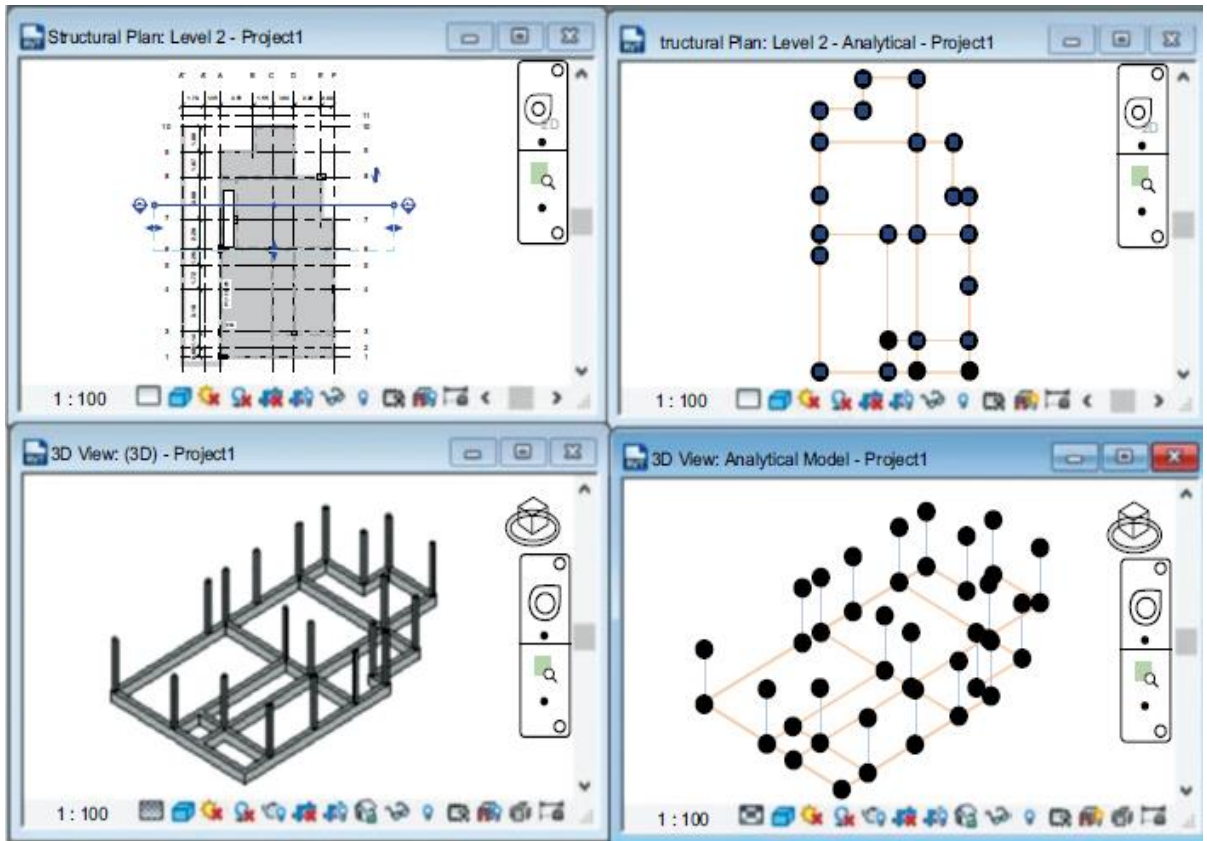


Fig. 3.11. Side-by-Side Representation of Geometric and Analytical Models

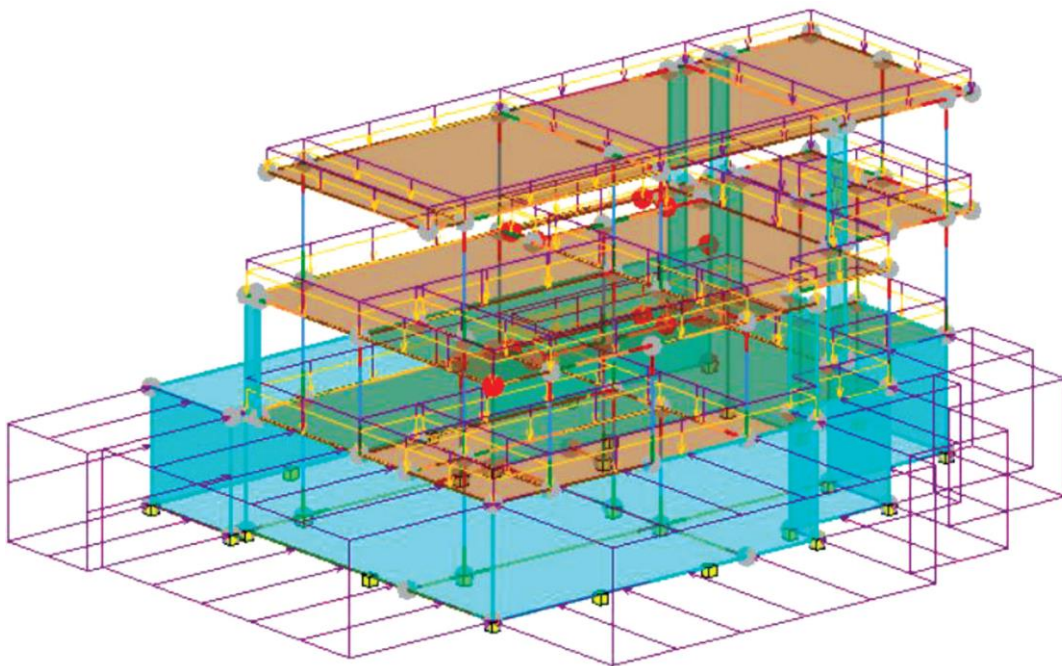


Fig. 3.12. Applying Loads

3.1.2. Model Transfer

The design of structures requires a structural calculation system. The structural model is transferred to an analysis tool in a BIM workflow, often using the universal Industry Foundation Classes (IFC) format. However, IFC transfer can cause interoperability issues, leading to errors, omissions, and data duplication.

According to the study by Sampaio and Gomes [3], the model created in Revit was transferred to Robot, an Autodesk analysis tool, using the native format, which ensures better interoperability. Despite this, some errors were detected during the transfer, mainly due to regional differences and parameter inconsistencies:

1. Concrete C30/37 kept its designation, but mechanical property values needed correction.
2. Section dimensions and thicknesses were correct, but element positions required adjustment; upper surfaces of beams and slabs had to be leveled, except on the top floor.
3. Loads were transferred with corrections, but load combinations and safety factors did not fully match the Last Limit State (LLS) and Service Limit State (SLS).

After correcting these issues, the structural analysis was carried out, efforts were calculated, pre-dimensioning was verified, and reinforcement detailing was established.

4. STRUCTURAL ANALYSIS

The structural efforts were determined through a 3D analysis, assuming linear elastic material behavior and geometric linearity. Calculations followed applicable standards for loads, concrete design, and earthquake resistance.

Permanent loads include the self-weight of structural elements (77.5 kN/m^3 for concrete, 55 kN/m^3 for steel), non-structural walls, coatings, and soil pressure on foundation walls (20 kN/m^2). Variable loads account for normative use: 2 kN/m^2 on floors 0 and 1, and 1 kN/m^2 on the roof. Seismic loads consider the building's location in Algarve, seismic zones 1.3 and 2.3, accelerations of 1.5 m/s^2 and 1.7 m/s^2 , and soil type B.

Action combinations were defined according to current standards for Ultimate Limit State (ULS) and Service Limit State (SLS). Structural checks include deformation and cracking, with minimum reinforcement requirements, spacing, and diameters controlled. Deflection limits are $1/250$ for solid slabs and beams under near-permanent loads, and $1/500$ in partition wall zones.

Permanent and variable loads were correctly applied, while safety factors in combinations were adjusted and seismic action added. Robot facilitates easy load application and combination definition, so importing the model without loads and applying them directly within Robot is recommended.

4.1. Structural Analysis Results

The analysis of structural efforts confirms the adequacy of the initially established pre-sizing. Efforts are presented both for individual load combinations and as envelopes of all combinations, allowing verification of the structure under ULS and SLS conditions. Stresses and deformations are visualized through 2D diagrams or 3D models (see Figs. 4.1, 4.2).

The software performs a dynamic analysis for seismic loads, providing results as numerical tables and dynamic animations showing the structure's vibration modes (see Fig. 4.3, Table 4.1).

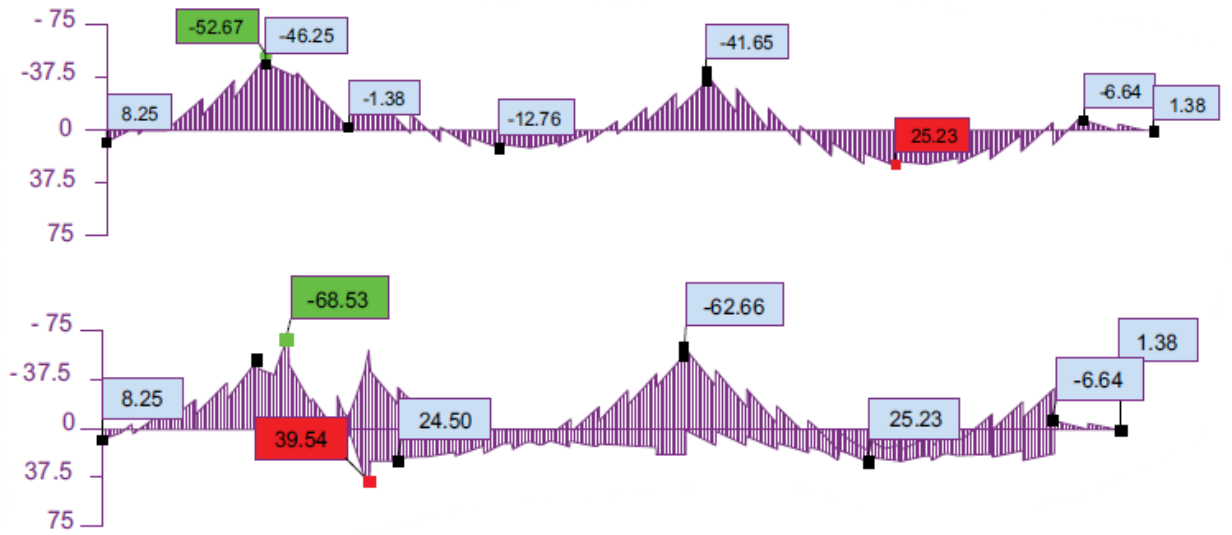


Fig. 4.1. Bending Moment Diagrams for Beams

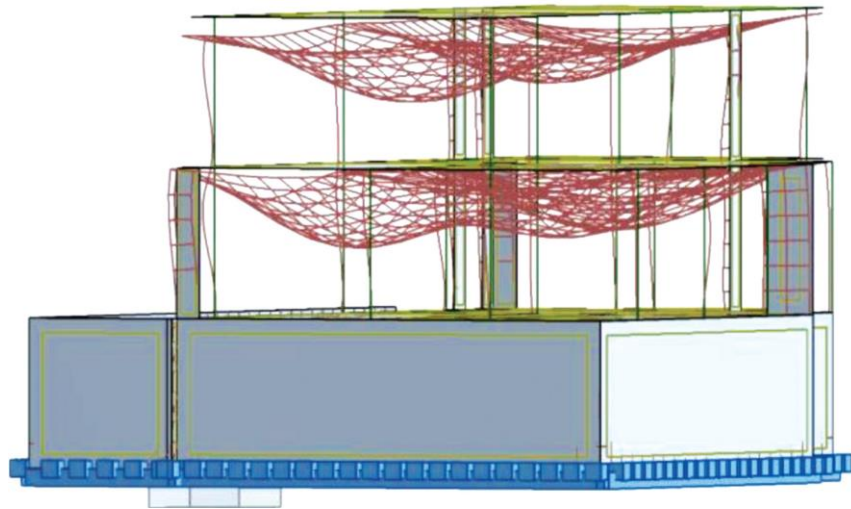


Fig. 4.2. Slab Deformation Diagrams

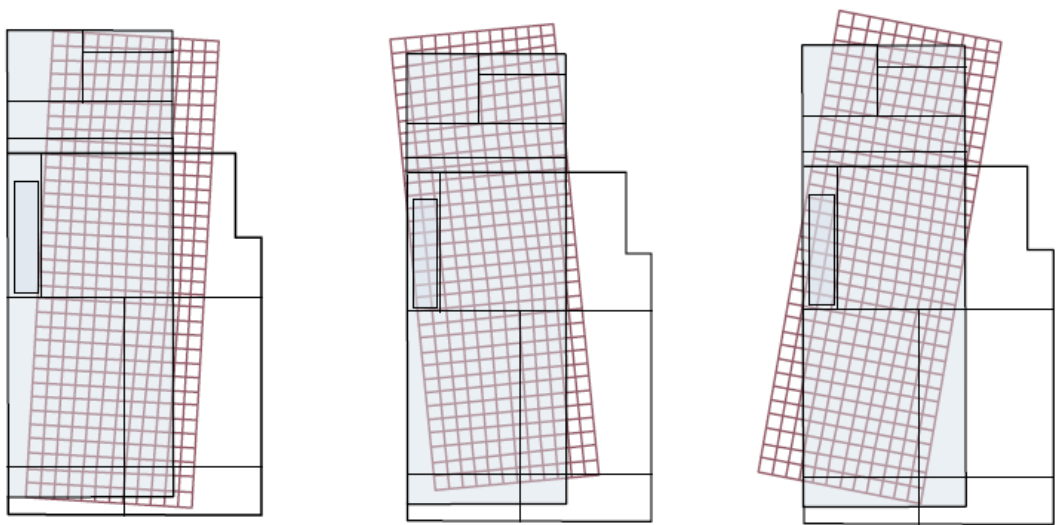


Fig. 4.3. Modes of Vibration

Model Displacements in X and Y

Mode	Frequency (H_z)	Period (s)	Displacement in X		Displacement in Y	
			UX (%)	SUX (%)	UY (%)	SUY (%)
1	3.28	0.30	46.77	46.77	3.70	3.70
2	3.70	0.27	16.58	63.35	23.75	24.45
3	4.43	0.23	0.89	64.24	55.69	83.14
4	8.07	0.12	34.86	99.10	0.00	83.14
5	8.42	0.12	0.00	99.10	6.35	83.49
6	12.19	0.08	0.04	99.14	9.70	99.18

4.2. Reinforcement Detailing

The required reinforcement areas for each member are determined based on the calculated efforts. The software enables automatic reinforcement detailing, controlled by designer-defined parameters. Figs. 4.4 and 4.5 show the reinforcement detail for beam VA on floor 0, generated using the system interface and the calculated reinforcement area (see Table 4.2).

Reinforcement for basement column P1 is detailed based on the compound bending effort (see Figs. 4.6, 4.7). Using the software's capabilities, foundation and slab sizing can also be performed, with results presented as tables, diagrams, or calculation notes (see Figs. 4.8, 4.9, and Table 4.3).

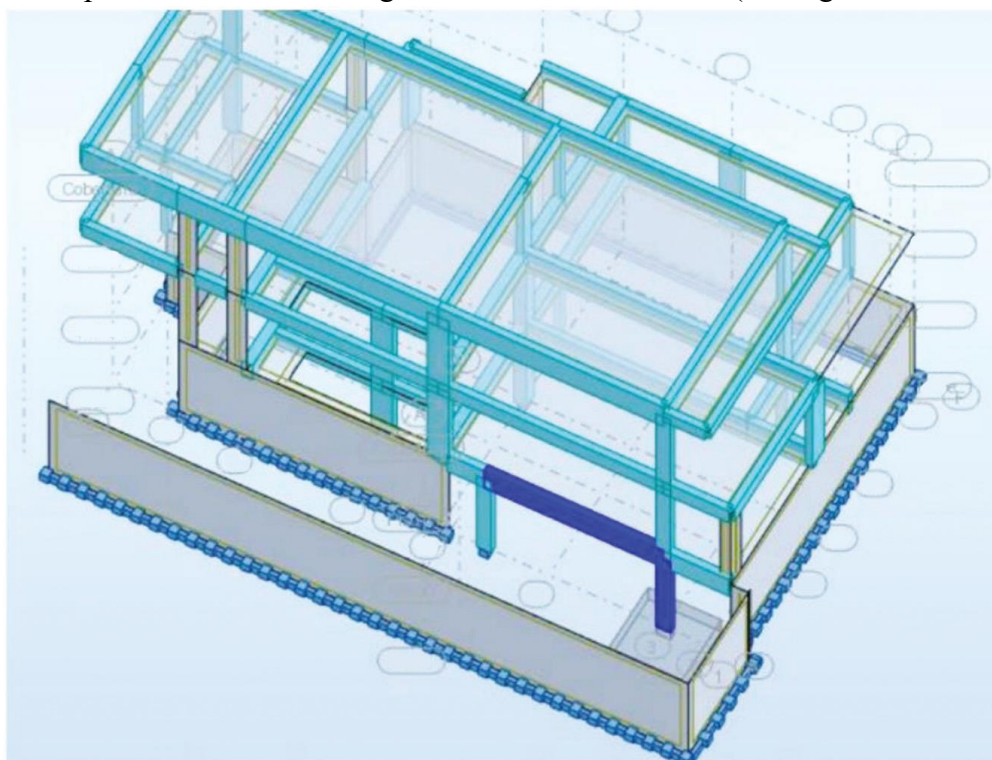


Fig. 4.4. Beam Selection within the Model

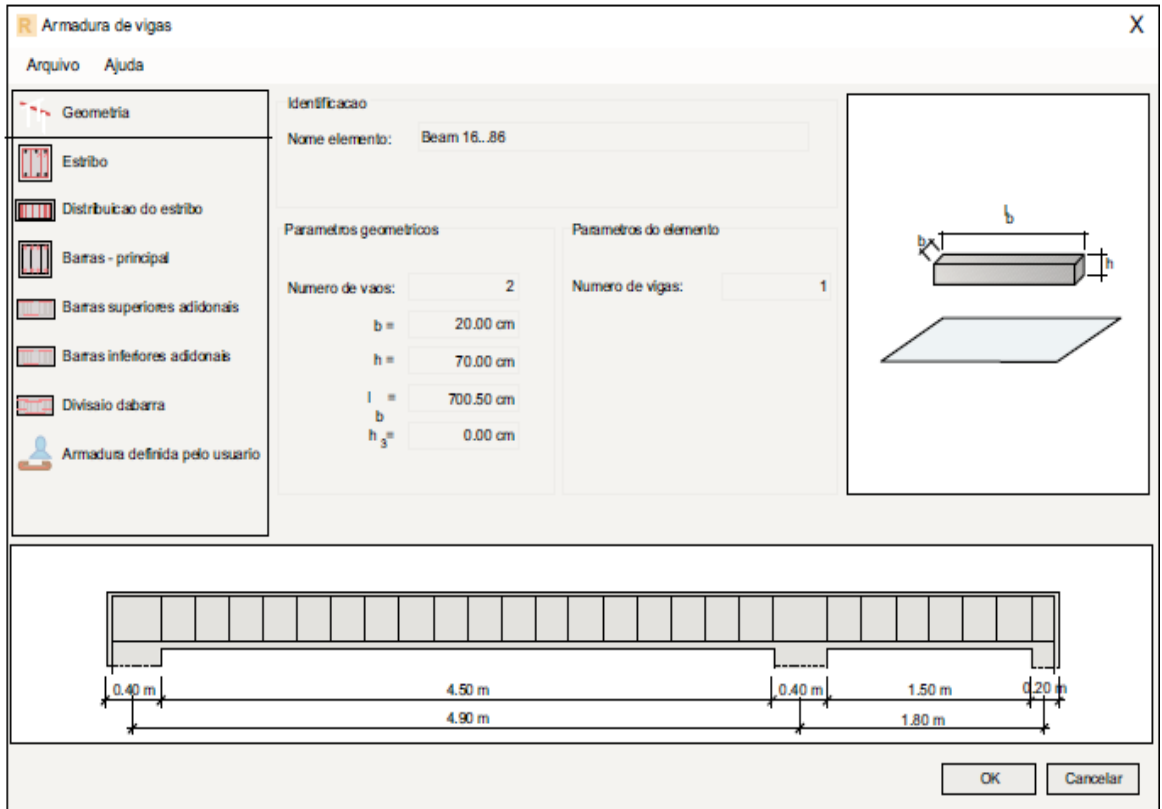


Fig. 4.5. Beam Reinforcement Design Interface

Table 4.2.

Required Reinforcement Areas for a Beam

M (kN×m)	M_{ed} (kN×m)	A_s (cm ²) nec.	Solution	b (m)	d (m)	f_{cd} (kPa)	f_{yd} (MPa)
M^{+}_{min}	24.94	0.88	2F16	0.20	0.66	20000	434782.6
M^{-}_{max}	32.60	1.15					

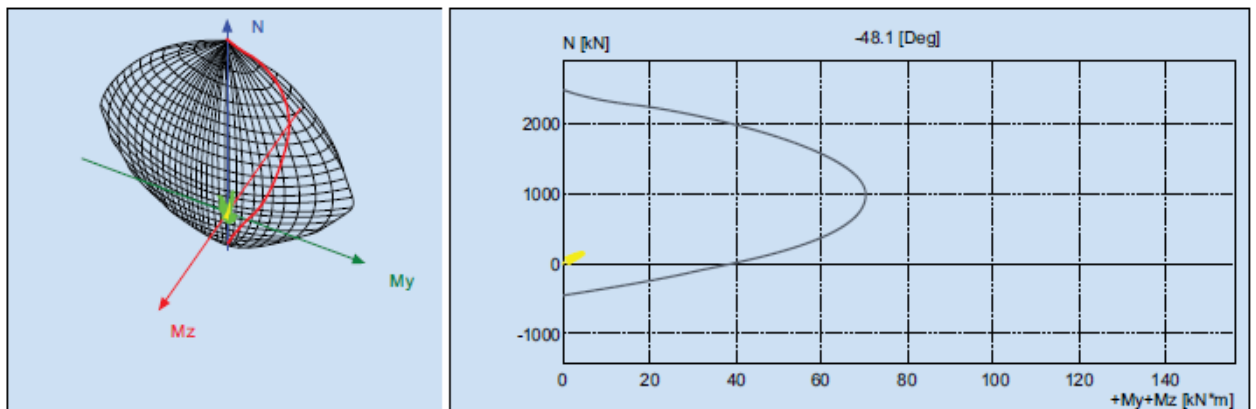


Fig. 4.6. Column N-M Interaction Diagram

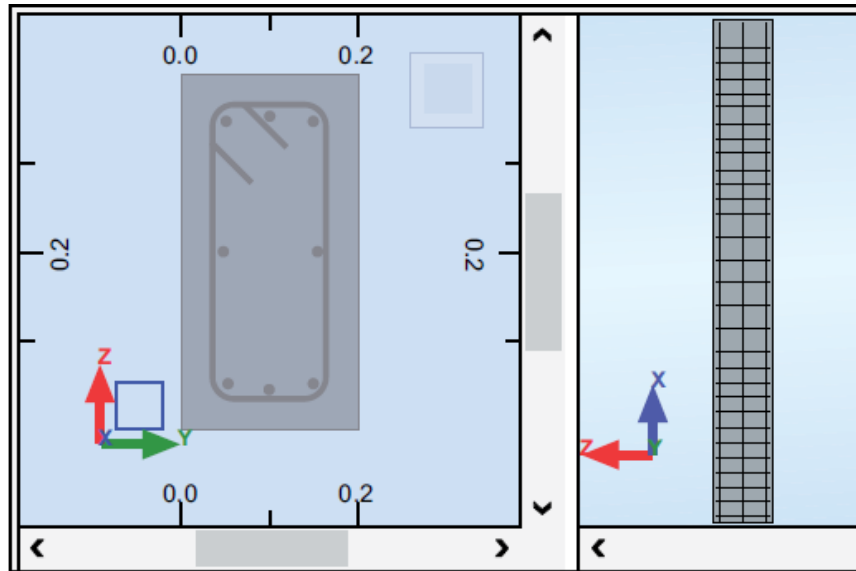


Fig. 4.7. Column reinforcement representation

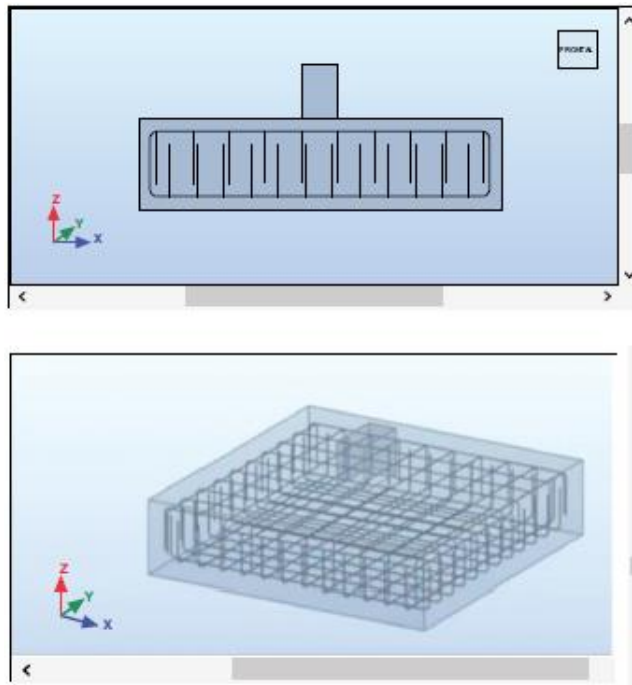


Fig. 4.8. Foundation Reinforcement Detailing

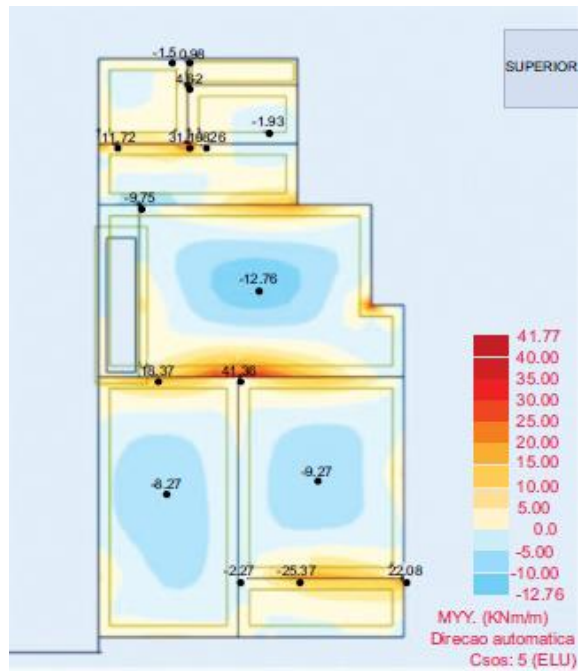


Fig. 4.9. Bending Moment Diagram for the 1st Floor Slab

Table 4.3.

Reinforcement Areas in Foundations

Foundations	$A_{s,int}$ (Solution)	$A_{s,int}$ (cm ² /m)	$A_{s,sup}$ (Solution)	$A_{s,sup}$ (cm ² /m)
Direction $x-x$	F12//0.15	7.54	F10//0.20	3.93
Direction $y-y$				

4.3. Reinforcement Information Transfer

BIM methodology relies on centralizing project information on a single platform, including geometric models, drawings, tables, and analysis reports. Structural analysis results must be transferred back to the initial model. This phase is often problematic due to remaining inaccuracies, despite technological advances and improvements in the IFC format.

When transferring reinforcement data from Robot to Revit, some inconsistencies occur. For example, continuous beams in Robot spanning multiple adjacent spans are modeled individually in Revit, causing reinforcement to be repeated for each span. This is corrected by removing excess reinforcement elements (see Fig. 4.10).

Revit did not identify reinforcements in foundations because the initial transfer treated foundations as simple supports without dimensions. Similarly, slab reinforcement data from Robot is not recognized in Revit. To resolve these issues, reinforcement detailing must be completed or corrected using Revit tools and Extensions Reinforcement (see Figs. 4.11–4.13), which allow intuitive, partially automated reinforcement generation and copying between similar members. The robot also generated detailed drawings to support modeling and graphic documentation (see Fig. 4.14).

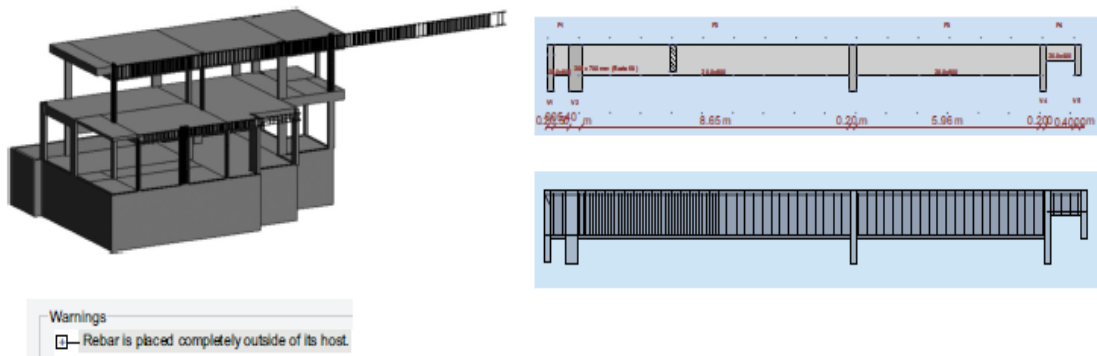


Fig. 4.10. Error in Revit Transfer Resulting from a Beam Defined as Continuous across All Spans in Robot

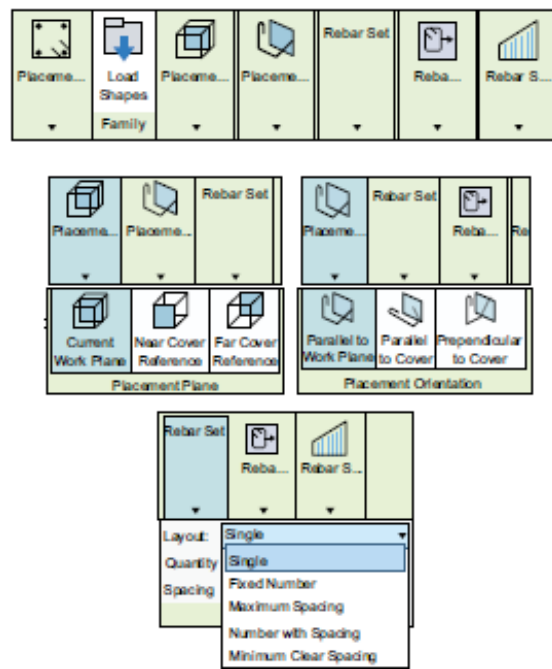


Fig. 4.11. Interfaces of Revit Reinforcement Extensions: Element Types

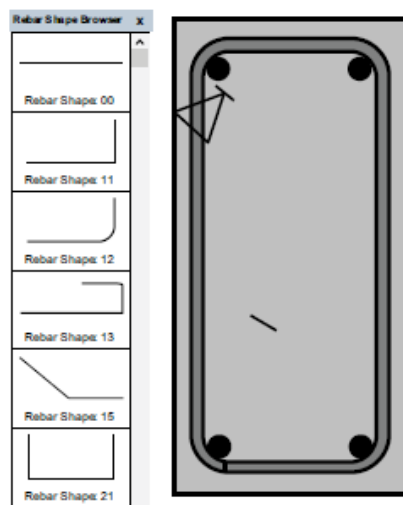


Fig. 4.12. Interfaces of Revit Reinforcement Extensions: Beam

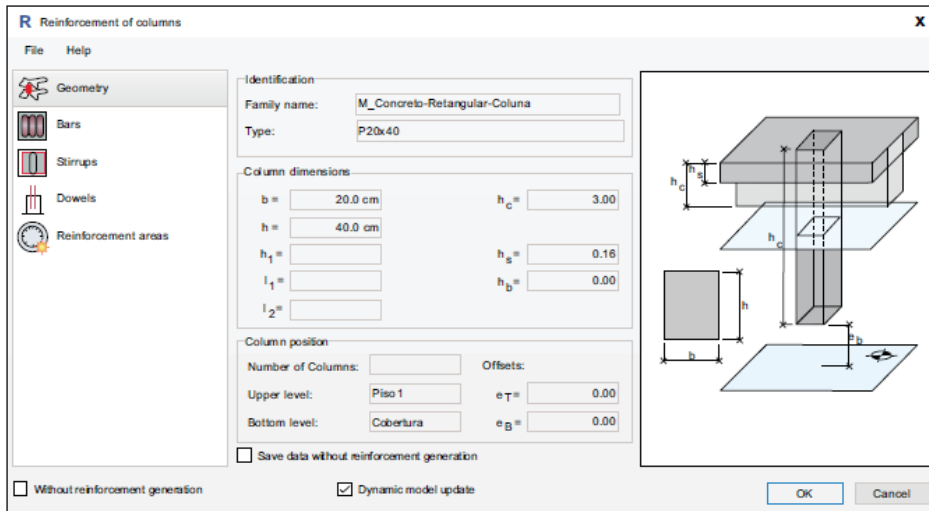
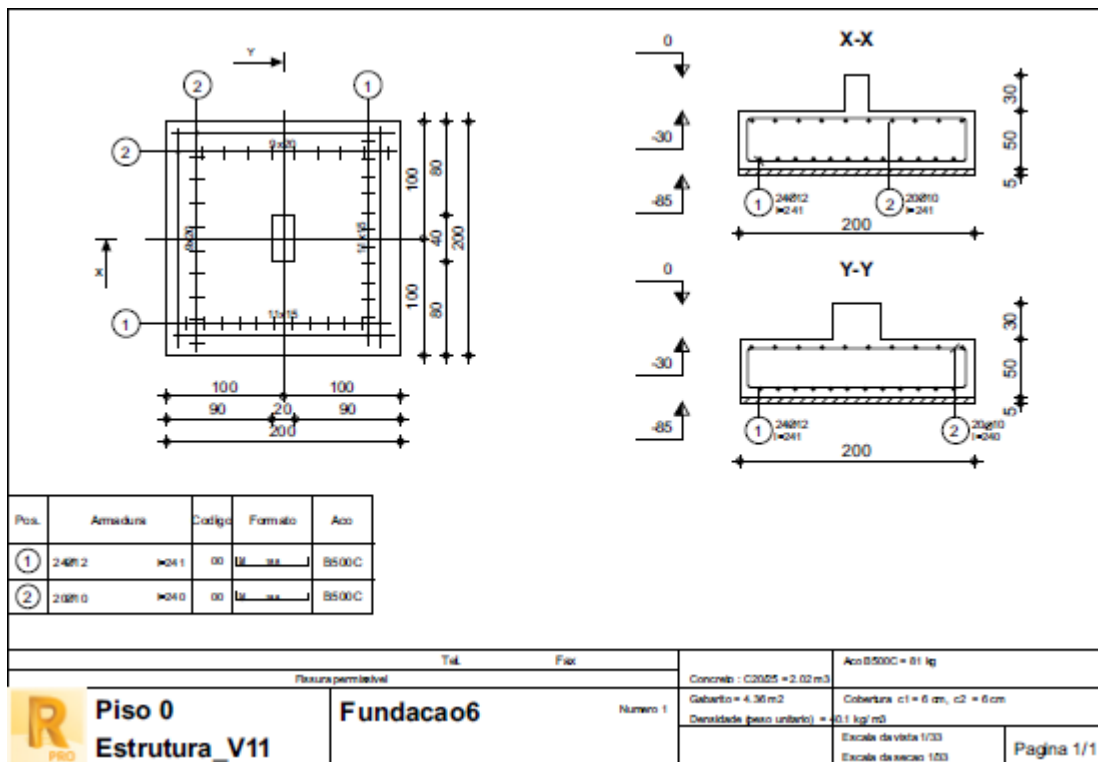


Fig. 4.13. Interfaces of Revit Reinforcement Extensions: Columns



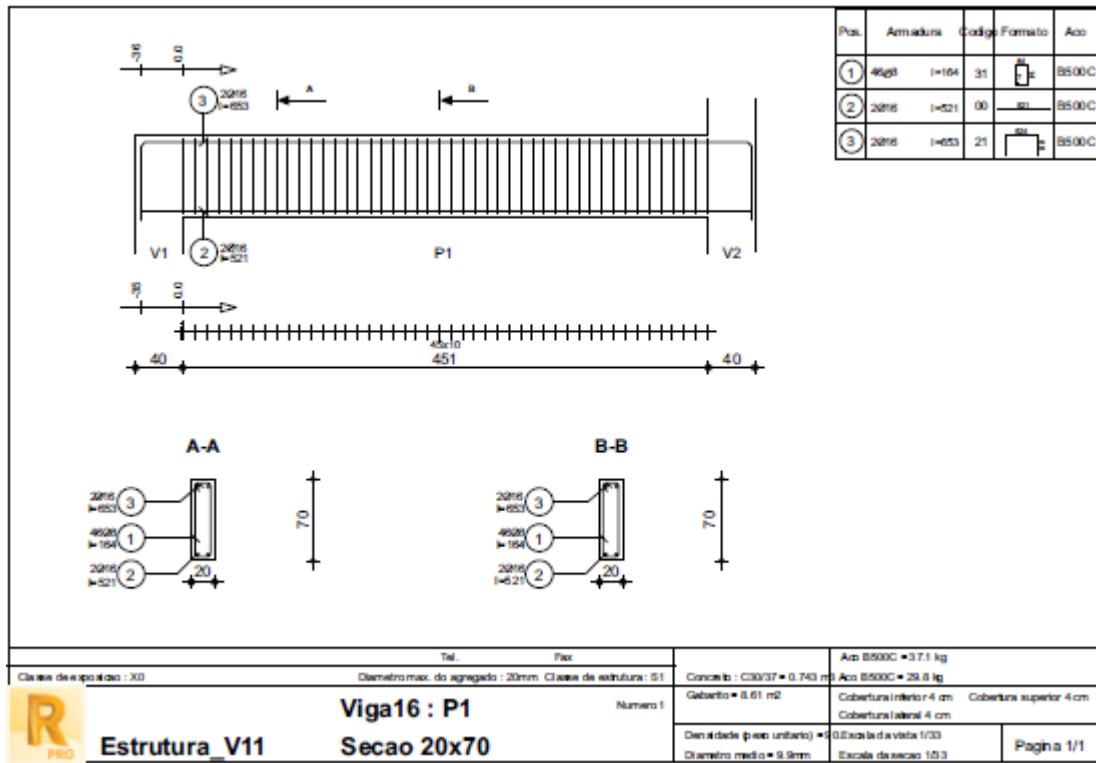


Fig. 4.14. Drawing Sheets Generated Automatically by Robot

These representations were saved in DWG format in Robot and imported into Revit to model the missing reinforcements. With reinforcement modeling on all elements, the BIM model becomes complete. Traditional project outputs like drawings and result tables can be generated easily using Revit's built-in features.

4.4. Drawings and Reinforcement Quantities

Drawings and material quantity tables are generated from the complete, updated model. BIM tools allow dynamic creation and archiving of projections, sections, and tables directly from the model. As the project evolves with changes and adjustments, these drawings and tables are automatically updated. This ensures consistency and enables efficient extraction of information for project documentation, budgeting, maintenance planning, or building management.

4.5. Graphic Documents

Reinforcements were modeled for all members using the Extensions Reinforcement tool, enabling the generation of drawings and tables. While transferring graphics from Robot to Revit still requires some work, much of the documentation is produced automatically in Robot and imported into Revit. This demonstrates that BIM tools significantly streamline the creation of technical drawings for structural designers. The geometric model serves as the basis for drawings, eliminating manual generation of sections, elevations, and plans. Drawings correspond to views or perspectives applied to the model and are stored on the platform as dynamic, individual schemas (see Figs. 4.15, 4.16).

Revit uses families to manage members such as lines, text, and dimensions (see Figs. 4.17, 4.18). Annotations in each drawing can be customized to match standard structural documentation. However, some user experience is needed for dynamic block adjustments.

Revit automatically identifies reinforcements when showing a beam cross-section, but only one longitudinal bar and a single stirrup are annotated. Usually, all longitudinal bars and stirrup

spacing should be labeled (see Figs. 4.9, 4.20). Additional annotation work is required, but updates to the cross-section automatically adjust the labels.

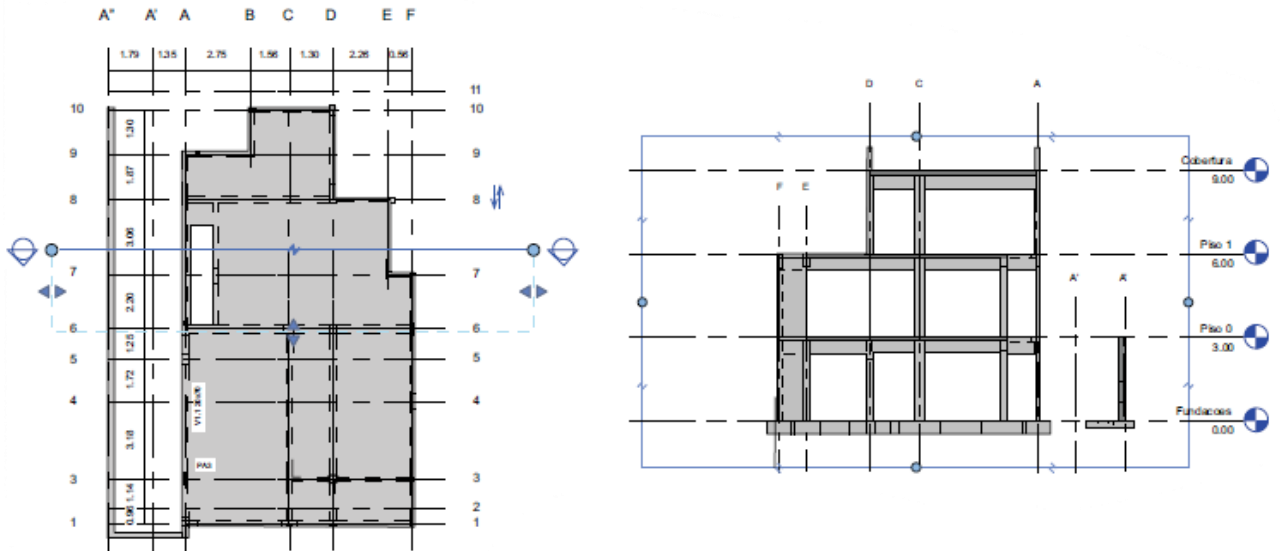


Fig. 4.15. Vertical Section Generated on the 2nd Floor Plan

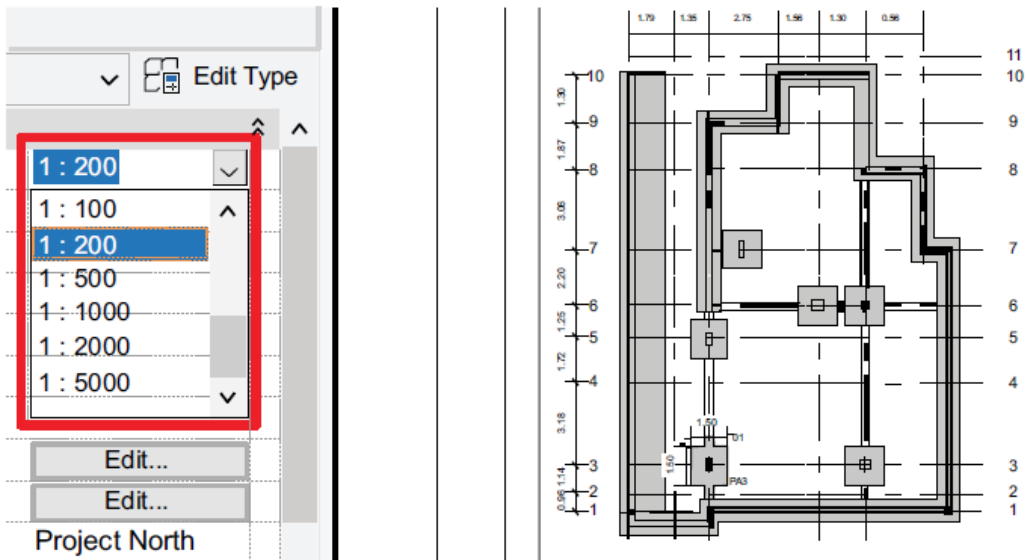


Fig. 4.16. Choosing a Scale for Displaying Foundation Drawings

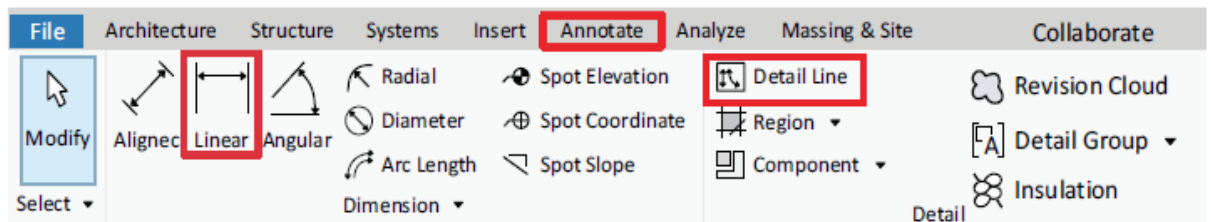


Fig. 4.17. Dimension Tools Menu

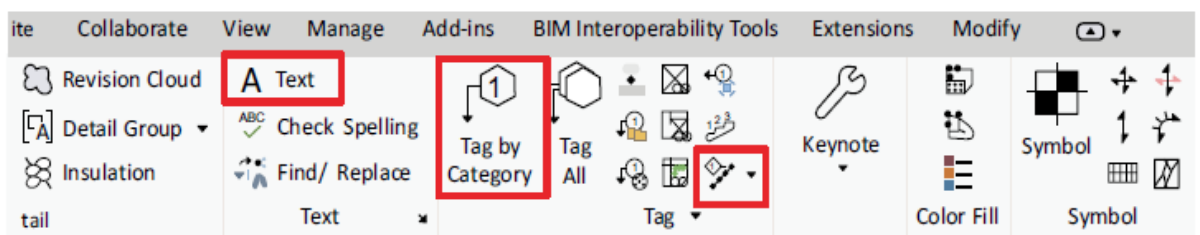


Fig. 4.18. Caption Annotations

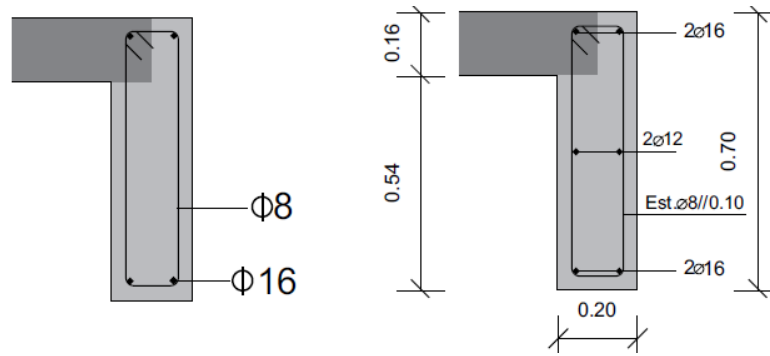


Fig. 4.19. Automatic (Left) and Manual (Right) Reinforcement Detailing in a Beam

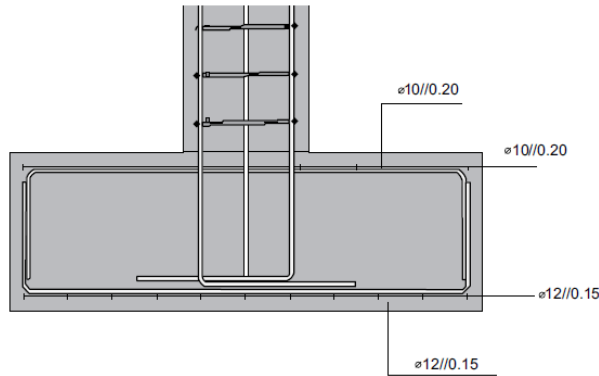


Fig. 4.20. Manual Reinforcement Detailing in a Foundation

				<p>Projeto de estruturas apoiado na metodologia BIM: análise estrutural e pormenorização de armaduras</p>
				<p>Anexo 7</p> <p>Plantas e Alçados</p> <p>Data: Outubro 2019</p> <p>Desenhada por: Nuno Borges</p>

Fig. 4.21. Arrangement of Drawing Sheets with Graphical Layouts and Dynamic Frames

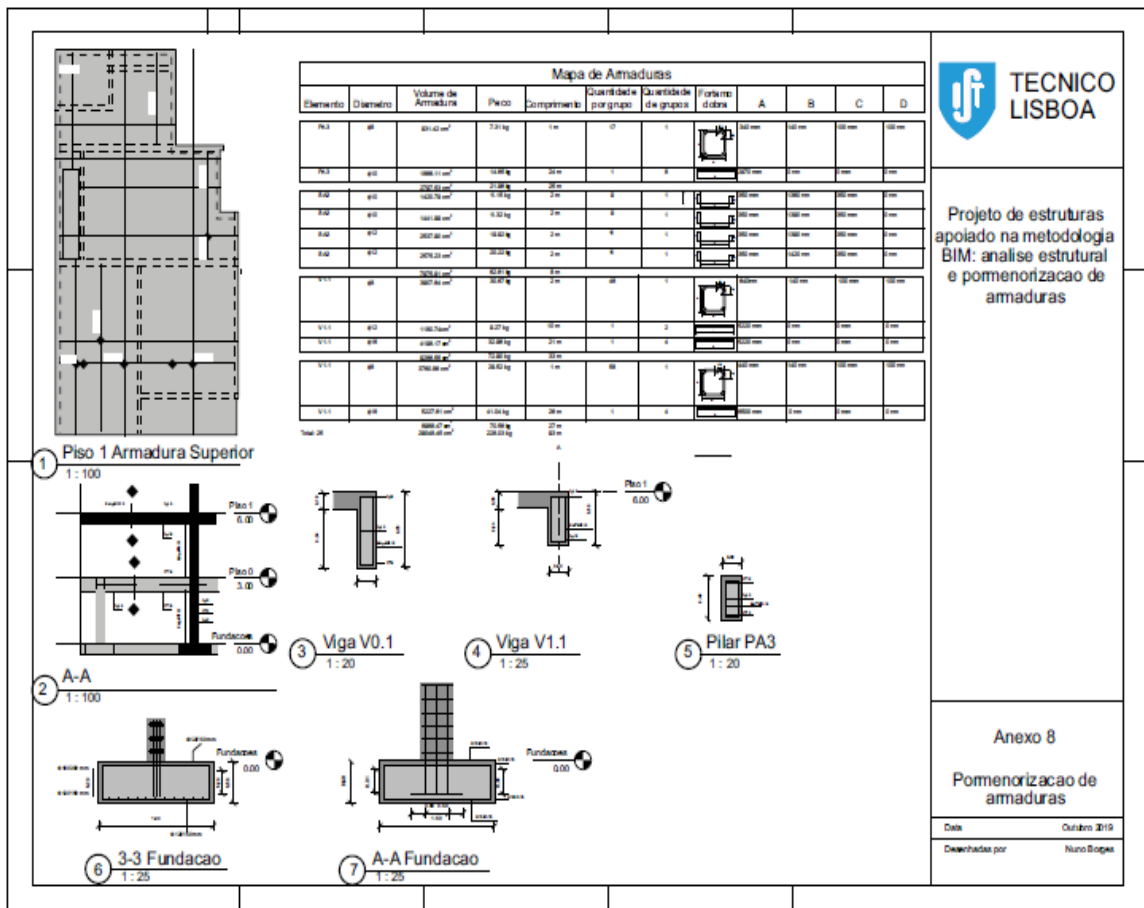


Fig. 4.21 (continued). Arrangement of Drawing Sheets with Graphical Layouts and Dynamic Frames

Different drawings and tables can be compiled into drawing sheets linked to the project. Several sheets were created to achieve the desired graphics and structural presentation (see Fig. 4.21).

4.6. Quantity Take-off

In BIM, calculation data can be accessed by engineers and integrated into a single project platform. Quantity tables (QTO) list materials, element sizes, mechanical properties, and load values. QTO provides detailed measurements of materials and labor for project completion. Calculation notes from Robot cannot be attached directly to Revit but can be exported as images or to Excel/Word for reference, though updates must be done in Robot first.

For seismic or modal analyses, Revit only identifies loads and combinations; calculation results from Robot can be added as images, including stress and deformation diagrams (see Figs. 4.22, 4.23).

Material quantities, including concrete volume and reinforcement per element or floor, can be organized in dynamic tables with associated unit costs and labor. Revit allows editable tables to be included in drawings, exported to Excel, or used in project documentation.

Finally, reinforcement format maps showing element dimensions and quantities can be generated. These maps aid construction by providing cutting and bending details or supporting factory prefabrication, reducing on-site space and assembly time.

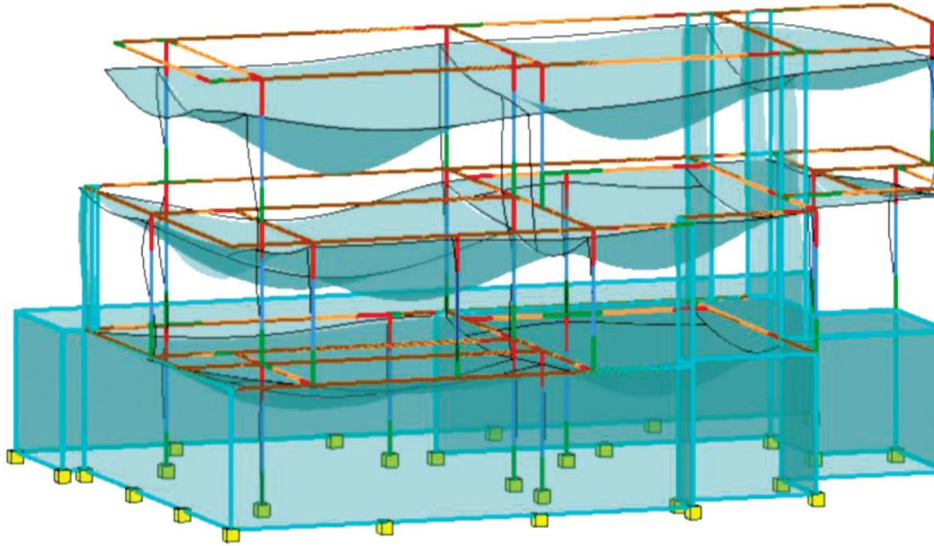


Fig. 4.22. Slab Deformations

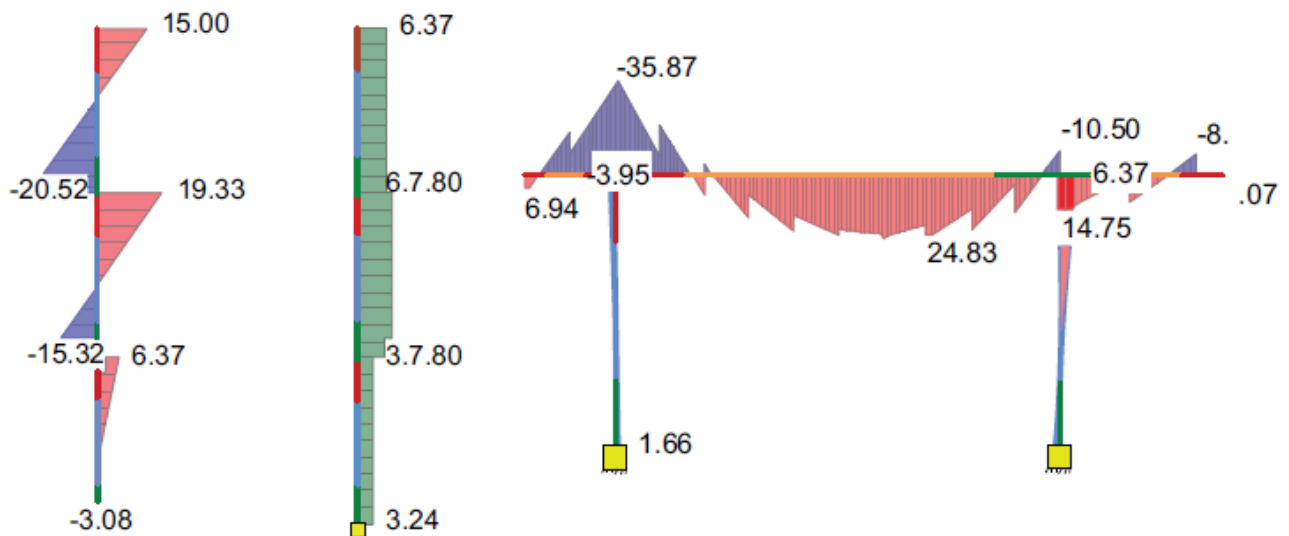


Fig. 4.23. Column (Left) and Beam (Right) Stress Diagrams

5. DISCUSSION

The analyzed study highlights BIM's key advantages and limitations for structural engineering.

5.1. Benefits identified:

1. High parametric modeling capacity allows for exploring alternative solutions and verifying the model geometrically and analytically before analysis.
2. Revit–Robot transfer enables error checking and fixes.
3. Dynamic drawings and tables are automatically updated from the final model; drawing sheets can include projections, diagrams, and tables.
4. Reinforcement tables and maps showing rod diameters, quantities, and configuration support on-site preparation and factory prefabrication.

5.2. Limitations observed:

1. Geometric consistency must be carefully verified in Revit before and after transferring to Robot.
2. Transferring loads, combinations, and seismic actions between Revit and Robot can be problematic.
3. Reinforcements from Robot often contain errors when imported into Revit.

4. Notes, diagrams, and animations must be imported as static images, with no automatic updates.

5.3. Recommendations for engineers:

1. Define, optimize, and verify the structural model in Revit.
2. Apply all loads and combinations, including seismic, in Robot.
3. Generate drawings, models, graphics, and reports in Robot and then transfer to Revit, correcting any lost elements.
4. Use Revit's dynamic tables, drawings, and maps to ensure consistency and easy updates.

6. CONCLUSIONS

1. The BIM methodology offers significant advantages across various construction areas and is experiencing rapid global adoption. Using basic BIM tools in structural design enables efficient parametric modeling, easy verification of the analytical model's consistency, and partial automation of creating graphical documents.

2. Interoperability issues can arise in the two-way data exchange between modeling and analysis systems, and implementing BIM in structural design requires additional training and mastery of extensions.

3. The workflow includes the whole cycle: creating a model in Revit, analyzing it in Robot, transferring results back to Revit while correcting limitations, and preparing drawings and quantity maps. This allows defining the most effective strategy for applying BIM in structural offices.

4. Overall, BIM implementation speeds up and automates the structural design process. Engineers must clearly understand which tasks are more efficiently performed in each system and which additional procedures should be applied.

7. REFERENCES

1. Borges N.M. Design of structures supported by BIM methodology: structural analysis and reinforcement detailing. MS Thesis. Lisbon, Portugal: Technical University. 2019.

2. Sacks R. et al. BIM Handbook: A guide to building information modeling for owners, designers, engineers, contractors, and facility managers, 3rd ed. *John Wiley & Sons, Inc.* 2018. 688 pgs. <http://dx.doi.org/10.1002/9781119287568>.

3. Sampaio A.Z., Gomes A.M. BIM Structural Project Applied in A Case Study: Interoperability Analyses, Reinforcement Detailing Drawings and Quantity Take-off. *The Open Construction & Building Technology Journal*. 2021. No. 15. P. 196–220. <https://doi.org/10.2174/1874836802115010196>.

4. Underwood J. and Isikdag U. Handbook of research on building information modeling and construction informatics: Concepts and technologies. 2009. 757 pgs. <http://dx.doi.org/10.4018/978-1-60566-928-1>.

5. Wang Q. et al. Cell-Based Transport Path Obstruction Detection Approach for 4D BIM Construction Planning. *Journal of Construction Engineering and Management*. 2018. No. 145(3). [http://dx.doi.org/10.1061/\(ASCE\)CO.1943-7862.0001583](http://dx.doi.org/10.1061/(ASCE)CO.1943-7862.0001583).

Educational publication

Bolotov Maksym Hennadiiovych
Hanieiev Timur Rashytovych
Davydov Ihor Ihorovych
Zahilskyi Vitalii Anatoliiovych
Kovba Vladyslav Valeriiovych
Levkivskyi Dmytro Volodymyrovych
Malakhov Viktor Valeriiovych
Moskvitina Anna Serhiivna
Pozniak Oksana Romanivna
Pushchina Nataliia Valentynivna
Rybachov Serhii Hryhorovych
Rudenko Mykhailo Mykolaiovych
Shekhovtsov Vladyslav Ihorovych
Siedin Volodymyr Leonidovych
Soroka Mykola Mykolaiovych
Surianinov Mykola Heorhiiovych
Tytarenko Roman Yuriiiovych
Khmil Roman Yevhenovych
Chaban Viacheslav Petrovych

**MODERN APPROACHES TO THE DESIGN, ANALYSIS AND EVALUATION OF
BUILDING SYSTEMS USING BIM TECHNOLOGIES**

Case studies

E-edition

Editors: Yu.V. Bulhakova, N. S. Chernova

Computer layout: N. S. Chernova

Cover design: N. S. Chernova

Expert reviewer: I. I. Nazarenko, President of the Academy of Construction of Ukraine, Doctor of Technical Sciences, Professor, Department of Machines and Equipment of Technological Processes, Kyiv National University of Construction and Architecture, Honoured worker of Science and Technology of Ukraine (Kyiv)

Format: 1/16 Conventional printed sheets: 13,95. Publisher's sheets: 14,02.

Order No.: 121.

Publisher: Ukrainian State University of Science and Technologies

2 Lazaryana St., rooms 2216, 263 (Scientific library),

Dnipro, 49010, Ukraine

Publishing Certificate No. DK 7709 of December 14, 2022



Co-funded by
the European Union

The **Bridge**
IM architecture
engineering
construction

Case Study

MODERN APPROACHES to Design, Analysis and Assessment of Building Systems Using **BIM-Technologies**



<https://thebridgeproject.org.ua/>