People’s Democratic Republic of Algeria

Ministry of Higher Education and Scientific Research

8 May 1945 Guelma University

Faculty of Science and Technology

Department of Civil Engineering & Hydraulics

TP Computer-Aided Design

Intended for students in the 3rd year of Licence in Civil Engineering.

Elaborated by : Dr LAFIFI Brahim

Programme

Semestre : 6

Unité d'enseignement : UEM 6.1

Matière : Calcul assisté par ordinateur

VHS: 37h30 (TP: 2h30)

Crédits : 3

Coefficient : 2

Objectifs de l'enseignement :
Familiariser les étudiants aux logiciels de calcul en génie civil. L'étudiant doit connaitre les fonctionnalités essentielles d'un logiciel de calcul, en se basant sur un projet existant, et doit être capable de maitriser l'interface du logiciel et saisir correctement les données et récupérer les résultats.
Connaissances préalables recommandées :
Informatique 1 et 2 et informatique 3

Contenu de la matière :

Chapitre 1 : Concept de base sur les logiciels de calcul (3 semaines)
Mode de fonctionnement et méthodes de calcul utilisées, les logiciels fermés, les logiciels ouverts, avantages et limites des logiciels.

Chapitre 2 : Prise en main d'un logiciel disponible. (6 semaines)
Présentation de l'interface, l'environnement de travail, les données, les options, les résultats (numériques et graphiques), interprétation.

Chapitre 3 : Etude et suivi d'un projet réel (6 semaines)

Mode d'évaluation :
Contrôle continu : 100%

Références bibliographiques :

1. Manuel d'utilisation du logiciel hôte.

Program

Semester : 6

Teaching unit: UEM 6.1

Subject : Computer-aided design

VHS: 37h30 (TP: 2h30)

Credits : 3

Coefficient : 2

Teaching Objectives:
Familiarize students with civil engineering calculation software. The student must be acquainted with the essential functionalities of a calculation software, based on an existing project, and must be capable of mastering the software interface, correctly inputting data, and retrieving results.

Recommended prerequisite knowledge:
Computer Science 1 et 2 et Computer Science 3

Content of the subject:

Chapter 1 : Basic concept on calculation software (3 weeks)
Operating mode and calculation methods used, closed software, open software, advantages and limitations of software.

Chapter 2 : Getting started with Robot Structural Analysis software.

(6 weeks)

Presentation of the interface, the work environment, the data, the options, the results (numeric and graphical), interpretation.

Chapter 3 : Study and monitoring of a real project using RSA 2010.
(6 weeks)

Evaluation method:
Continuous assessment: 100%

Bibliographic references:
1. User Manual of the host software.

Table of contents

CHAPTER III : Study and monitoring of a real project using RSA 2010.

Chapter I

Basic concept on calculation software

I.1 Introduction

Calculation software, also known as computational software or mathematical software, is a type of computer program designed to perform mathematical and numerical calculations. It provides tools and algorithms for solving complex mathematical problems, performing data analysis, and simulating various mathematical models. It is utilized in various fields, including engineering, physics, chemistry, economics, finance, data analysis, and scientific research. It provides efficient and accurate methods to solve complex mathematical problems, analyze data, and model and simulate real-world phenomena, aiding professionals and researchers in their work.

Calculation software incorporates numerical methods to solve mathematical problems that cannot be solved analytically. These methods involve approximations and iterative algorithms to find numerical solutions for equations, optimization problems, differential equations, and other mathematical models. It often provides tools for statistical analysis and data visualization. Users can perform tasks such as data importing, cleaning, filtering, and statistical calculations like mean, median, standard deviation, regression analysis, and hypothesis testing. Visualization features allow for the creation of charts, graphs, and plots to better understand and present data.

Many calculation software packages provide programming and scripting languages that allow users to create custom algorithms, automate calculations, and extend the software's functionality. Users can write scripts or programs to perform repetitive tasks, implement specific algorithms, and integrate with other software or systems.

I.2 Fundamental Understanding of Calculation Software

Calculation software serves as a tool that assists engineers, designers, and analysts in performing complex mathematical computations and simulations related to various fields such as engineering, physics, finance, and more. These software applications are designed to automate calculations, reduce manual errors, and enhance efficiency in solving intricate problems.

It automates repetitive and time-consuming mathematical tasks, freeing professionals from manual calculations and allowing them to focus on analysis and decision-making.

It handles complex equations, formulas, and algorithms that may be challenging or time-consuming to solve manually. It performs calculations much faster and accurately than human calculations.

It ensures high precision and accuracy in numerical computations, reducing the risk of errors inherent in manual calculations.

Some software allows for iterative analysis, enabling engineers to refine designs, conduct sensitivity studies, and optimize solutions by easily modifying input parameters.

Many calculation software provides visual representations of data, results, and trends. Graphs, charts, and diagrams help users understand complex relationships and patterns within the data.

Some software offers simulation capabilities, allowing users to model real-world scenarios and predict outcomes based on various inputs and assumptions. It helps in analyzing and interpreting large datasets, identifying trends, outliers, and correlations within the data.

Complex calculations that would take hours or days to complete manually can be accomplished within seconds or minutes using calculation software.

Software tools typically generate reports and documentation detailing the calculations performed, methodologies used, and results obtained. This documentation is essential for validation, communication, and regulatory compliance.

I.3 Operation Mode and Calculation Methods Used

I.3.1 Operation Mode: This refers to how a software application functions and interacts with users. In the context of calculation software, the operation mode determines how users input data, initiate calculations, and interpret results.

I.3.2 Calculation Methods: These are the mathematical algorithms and techniques used by the software to perform calculations. Depending on the software's purpose, it may employ numerical methods, analytical methods, finite element methods, optimization algorithms, and more.

I.3.3 Closed-Source Software

a) Definition: Closed-source software, also known as proprietary software, is developed by a specific company or organization. Its source code is not publicly available, and users only have access to the compiled executable program.

b) Advantages: Closed-source software often comes with official support, regular updates, and comprehensive documentation. Companies have control over intellectual property and can maintain consistent user experiences.

c) Limitations: Users lack the ability to modify the software's source code, limiting customization and adaptability. Dependency on the software vendor for updates and bug fixes can pose challenges.

I.3.4 Open-Source Software

a) Definition: Open-source software has publicly available source code that can be freely accessed, modified, and redistributed by users. This collaborative approach encourages community contributions.

b) Advantages: Open-source software promotes collaboration, innovation, and customization. Users can modify the software to suit their needs and contribute improvements to the community.

c) Limitations: Support may be community-driven, leading to varying levels of assistance. Not all open-source projects receive regular updates or have comprehensive documentation.

I.4 Presentation of the Software Robot Structural Analysis (RSA)

Robot Structural Analysis is a software program used for structural analysis and design of buildings and other structures. It is developed by Autodesk and provides engineers and structural designers with advanced tools and capabilities to analyze the behavior and performance of various structural elements. The software incorporates finite element analysis (FEA) techniques to simulate and evaluate the structural response under different loading conditions, such as static, dynamic, and seismic loads. It enables users to perform comprehensive structural analysis, including calculations for forces, displacements, stresses, and deformations. With its intuitive user interface and extensive library of building codes and design standards, Robot Structural Analysis helps in optimizing the design and ensuring structural integrity and safety.

Robot allows users to create structural models using graphical tools. Users can define various types of structural elements, such as beams, columns, bracings, slabs, and walls. The modeling environment supports both 2D and 3D structures.

The software includes extensive libraries of materials and cross-sectional profiles, enabling accurate representation of real-world materials and components. Some additional details about the software are presented below:

1.4.1 Analysis Types: Robot Structural Analysis supports a wide range of analysis types, including linear and nonlinear static analysis, dynamic analysis, buckling analysis, and response spectrum analysis. These analysis capabilities allow engineers to assess the behavior of structures under different loading conditions and evaluate their stability and performance.

1.4.2 Structural Elements: The software offers a variety of predefined structural elements, such as beams, columns, slabs, walls, and foundations. Users can create and customize these elements to accurately model the structural components of their projects. It also supports advanced features like curved beams, pre-stressed elements, and composite structures.

1.4.3 Material Modeling: Robot Structural Analysis allows users to define material properties for different construction materials, including concrete, steel, timber, masonry, and more. The software incorporates material libraries with predefined properties based on international standards, simplifying the process of assigning material properties to structural elements.

1.4.4 Loads and Load Combinations: Engineers can apply various types of loads to the structural model, such as point loads, distributed loads, temperature loads, and wind loads. Robot Structural Analysis enables the creation and customization of load combinations according to design codes, considering different load cases and load factors.

1.4.5 Design Code Compliance: The software includes a comprehensive library of design codes and standards from around the world, allowing engineers to ensure compliance with local regulations and industry-specific requirements. By selecting the appropriate design code, Robot Structural Analysis performs automatic code checks and provides detailed reports on the structural elements' design status.

1.4.6 Advanced Analysis Features: Robot Structural Analysis offers advanced analysis features, such as nonlinear material behavior, large deformation analysis, soil-structure interaction, and time history analysis. These capabilities enable engineers to tackle complex structural problems and simulate real-world scenarios accurately.

1.4.7 Design Optimization and Exploration: The software provides tools for design optimization and exploration, allowing engineers to iteratively refine and improve their structural designs. Users can set design objectives and constraints, and the software automatically adjusts the structural elements' dimensions to find optimal solutions based on predefined criteria.

1.4.8 Integration with Other Software: Robot Structural Analysis can be integrated with other software applications, such as Autodesk Revit and AutoCAD. This integration facilitates seamless data exchange, enabling users to import structural models, collaborate with architects and other disciplines, and export analysis results for further documentation and detailing.

I.5. Definition of engineering problems using RSA

To ease the user's work, the RSA software features a comprehensive set of tools that simplify the analysis of structures:

- ✓ **The Concept of Objects**: In RSA, the creation of the structure model is carried out using typical construction objects: beams, columns, bracings, floors, walls. Thanks to this, during this stage of the study, structural elements acquire specific attributes of their own (including regulatory attributes). Thus, during the model definition stage, all regulatory parameters of the structure are set, enabling an immediate transition to regulatory analysis right after static calculations. The same applies to nodes. The traditional notion of nodes has lost its meaning as they are automatically defined during the creation of various objects.

- ✓ **Construction Lines** can be used as modeling aids.
- ✓ **A wide range of editing tools:** plane symmetry, translation, rotation, horizontal and vertical mirroring, division for a specific bar or a group of bars, intersection, etc.
- ✓ **Powerful Selection Tools**: Effective selection tools include mouse pointer selection, selection by attribute (section, thickness, etc.), window selection, capture selection, plane selection, and more.
- ✓ **Option to Define Custom Display Attributes**: Capability to define customized display attributes such as menus, toolbars, colors, fonts, views, and window arrangements.
- ✓ **Adding Dimensions to the Structure Model**
- ✓ **Automatic Verification of Model Consistency**: For instance, searching for instabilities, missing supports, isolated bars and nodes, etc.
- ✓ **Utilization of Configurable Standard Structure Libraries:** The Robot software comes equipped with a set of standard structural components that are commonly used in engineering projects. These components are pre-defined with standard properties and attributes, saving engineers time by eliminating the need to manually input these details for each individual element.
- ✓ **Automatic Component Labeling Capability:** The "Automatic Component Labeling Capability" in the Robot Structural Analysis software refers to the functionality that enables the software to automatically generate labels or identifiers for different components within a structural model. This feature simplifies the process of identifying and managing individual elements in a complex structural model, enhancing organization, communication, and analysis efficiency.
- ✓ **Capability to Create and Archive Parameters**: Ability to create and save parameters such as any material, elastic supports, and various types of loads. This feature allows engineers to define custom material properties beyond the built-in material libraries. Engineers can input specific material properties, such as elastic modulus, density, and Poisson's ratio, for materials that are not included in the default library. This is particularly useful when working with specialized or non-standard materials.
- ✓ **Quick Input Function** for creating climatic loads such as snow and wind, along with automatic weighting. The "Quick Input Function" streamlines the process of defining climatic loads, such as snow and wind loads, which are essential for structural analysis and design. Instead of manually inputting each load value, this feature offers a faster and more convenient way to specify these loads.

- ✓ **Climatic Load Parameters**: For snow and wind loads, engineers typically need to define parameters such as intensity, direction, exposure category, and more. The quick input function simplifies this process by offering a user-friendly interface that guides engineers through the required parameters.
- ✓ **Automatic Weighting**: The automatic weighting feature enhances accuracy by calculating loads based on predefined standards and codes. This ensures that the loads are applied correctly to the structure, considering factors such as location, building height, and other design-specific criteria.

- ✓ **Simultaneous Views**: The "Multi-Windowing Feature" allows users to work with multiple views or windows of the same model or different models simultaneously. This feature enhances productivity by providing a comprehensive overview and enabling users to perform various tasks concurrently.
- ✓ **Comparative Analysis**: Engineers often need to compare different analysis results, load combinations, or design variations. The multi-windowing feature enables side-by-side comparison of these views, aiding in making informed decisions and identifying differences.
- ✓ RSA allows opening multiple windows of the same type, which enables, for instance, in the graphic area, the simultaneous consultation of different objects, even those that are far apart, using separate windows with their own display settings (zoom, projection, etc.).
- ✓ **Work Areas**: At each stage of the structural analysis, the software window can comprise three distinct work areas:

  - o Graphical Definition Area (controlled with the mouse),
  - o Definition Dialog Box Area (interacted with through the keyboard),
  - o Spreadsheet Area containing all the objects defined up to that point for the given class.

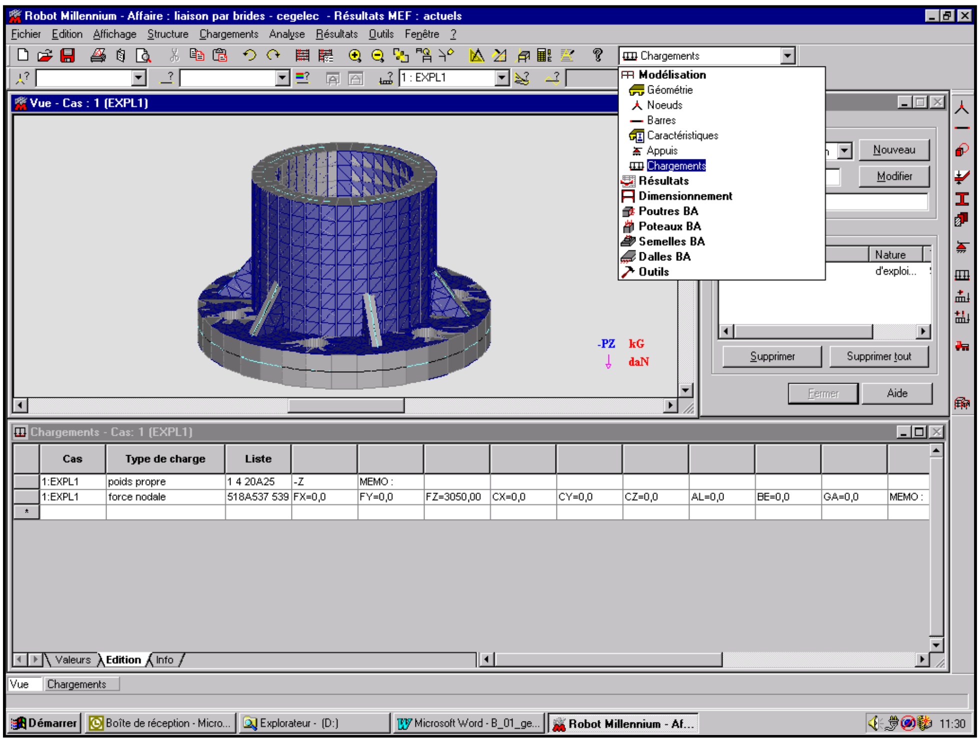


Figure I.1 Graphical interface of the RSA software.

I.6. Presentation of data and results

Data and results can be presented in both graphical and text modes: • Views of the structural model with node and bar numbers, support symbols, load diagrams with values, descriptions of sections used in the structure, drawings of the structure respecting the shape and

dimensions of sections, internal force diagrams, structural deformations, stress maps, displacements, and deformations for surface elements.
• Tables containing model descriptions.
• Tables displaying results.

I.6.1 Graphical presentation

- The software offers graphical representations of the structural model and analysis results. This visual presentation includes views of the structure, complete with labeled node and bar numbers for easy identification.
- Support symbols are displayed to indicate the location and type of supports in the structure.
- Load diagrams provide a visual representation of applied loads and their magnitudes. This helps engineers understand the load distribution and effects on the structure.
- Section descriptions are displayed, allowing users to quickly reference the specific sections used in the design.
- Drawings of the structure are generated, accurately representing the shape and dimensions of the sections used. This aids in visualizing the structure's physical appearance.

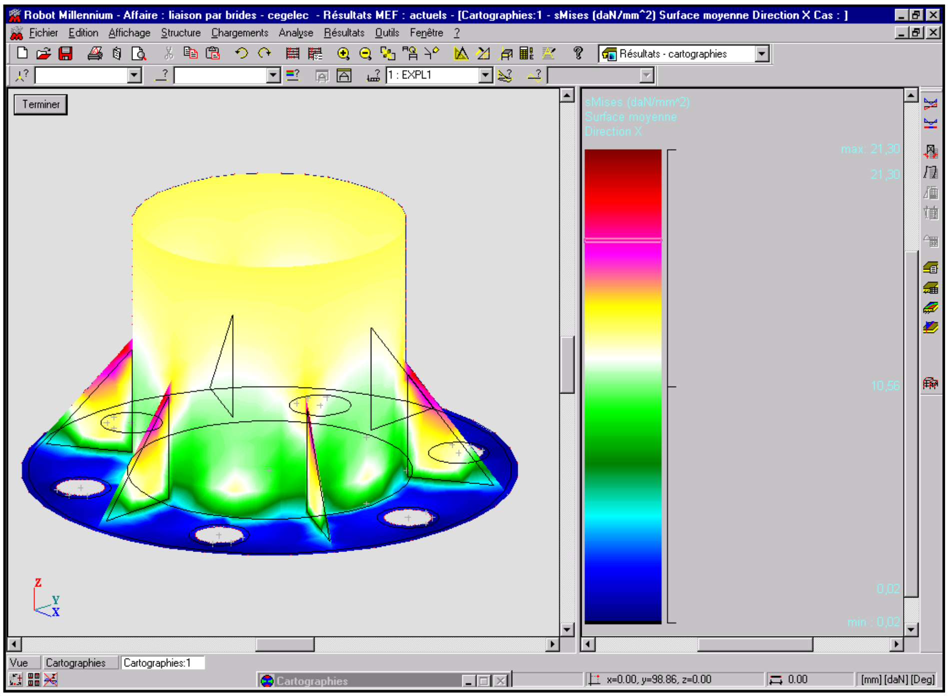


Figure I.2 Graphical representation of the results.

I.6.2 Tabular presentation

- In addition to graphical representations, the software provides tabular presentations of both model descriptions and analysis results.
- Model description tables list all the elements and attributes defined in the structural model, offering a comprehensive overview of the model's components.

- Result tables present numerical data obtained from the analysis. This includes values such as forces, displacements, reactions, and more.

I.7 Calculation notes

RSA software offers highly advanced tools for generating calculation notes. Thus, during the structural analysis, the content of the graphical screen or the active table can be captured at any moment. All the captured screens saved under a user-defined name can be inserted into the calculation notes. With this option, it is possible, for instance, to print the support table along with the structural plan featuring highlighted supports.

The "Print Composition" option available in RSA allows the user to freely compose the format and content of the calculation notes, including:

- Free composition of cover pages, headers, and footers (including graphic insertions).
- Defining the order of document elements to be printed.
- Composing the appearance of each page and table, even from the print preview.

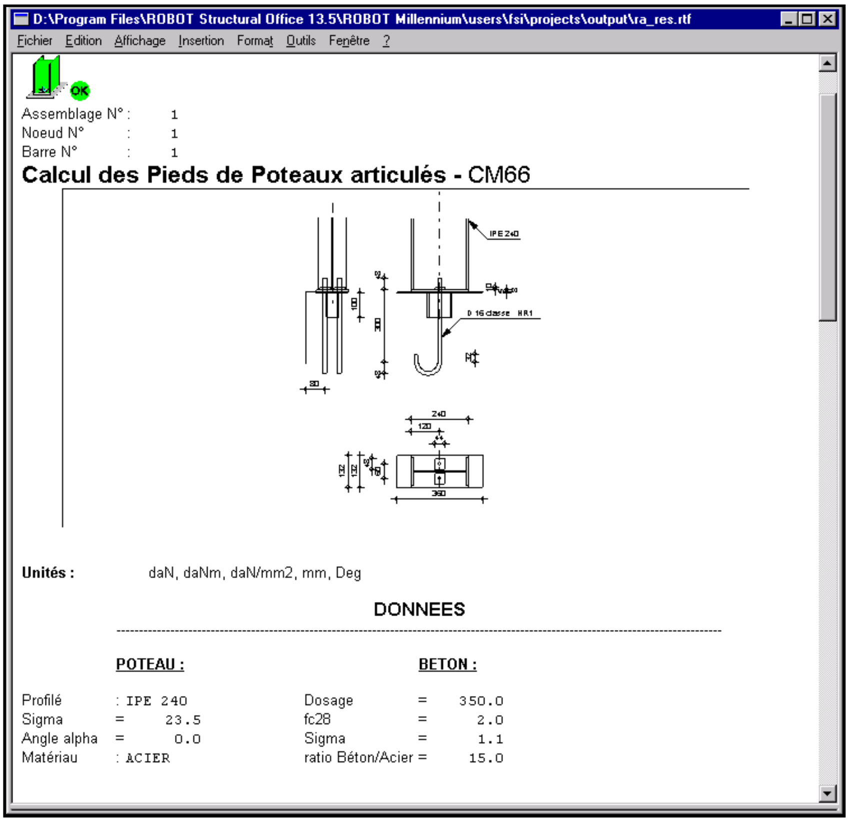


Figure I.3 Automatic generation of calculation notes.

I.8 Aid tools

A particularly crucial aspect for a powerful software like RSA is an assistance system that helps users master the system. Therefore, a significant effort has been made to provide effective aid tools, including:

- Contextual help for all menu commands and for each displayed object in dialog boxes and spreadsheets.
- Index of accessible help topics.
- Hierarchical access to information on the given subject.
- Descriptions of icons and menu commands displayed in the status bar at the bottom of the screen.
- Tooltips displaying the names of icons when the mouse pointer is placed over them.
- Similar tooltips accompany the mouse pointer during graphical input of structural elements. Their function is to inform the user of the effect that clicking the left mouse button can produce (for example, inputting the origin or end of a bar).
- The CD-ROM contains the complete "User Manual" and the "Quick Start Guide," which step-by-step presents the process of defining various structures (with explanations).

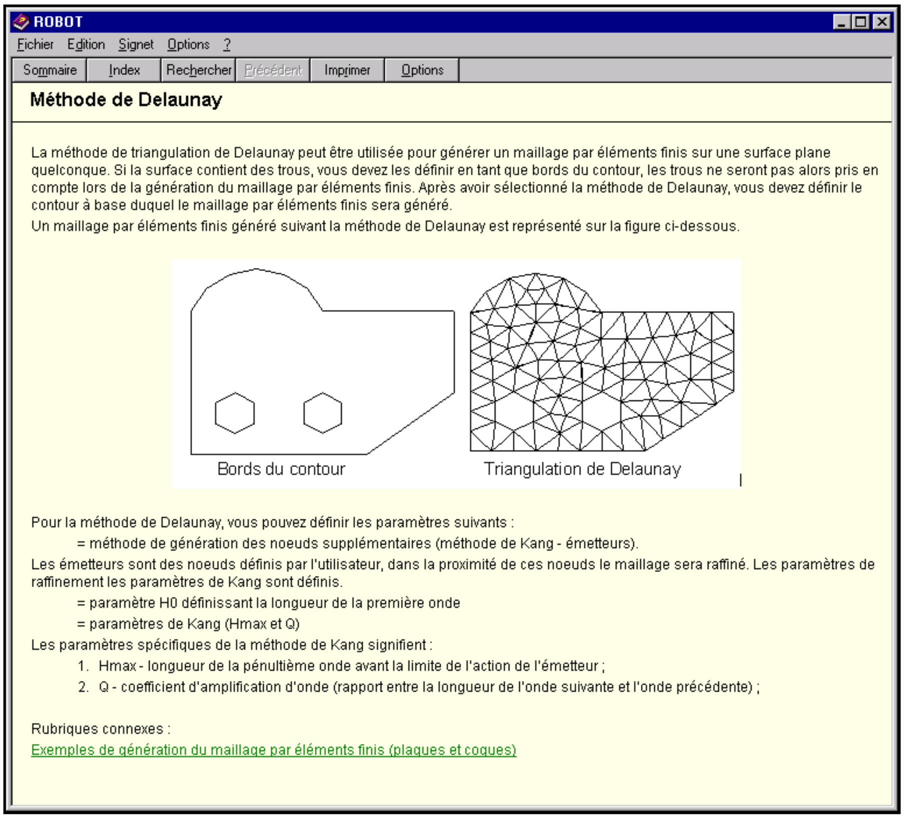


Figure I.4 Software help section.

Chapter II

Getting started with Robot Structural Analysis software

II.1 Launching the Robot Structural Analysis software

At the software startup, the following window appears to select the type of structure or element you want to study.

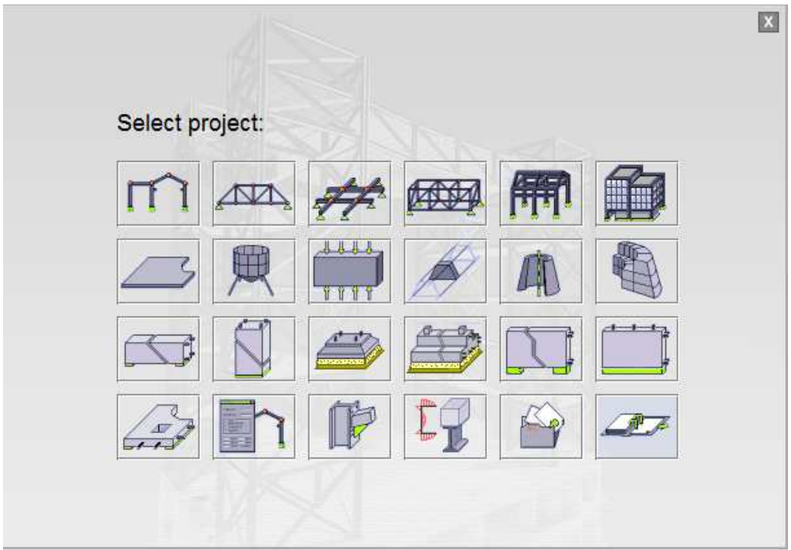
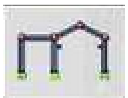
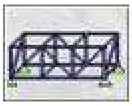
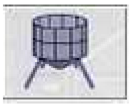
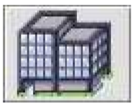
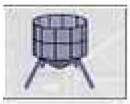


Figure II.1: Various Applications of Robot Structural Analysis software.

To facilitate modeling, they have included several modules to choose from, such as 2D or 3D options like Frames.

-  Analysis of a planar portal frame.
-  Analysis of a spatial truss.
-  Analysis of a shell.
-  Design of a building.

You can bring up this window at any time by clicking on the File menu, then selecting New Project. From there, you will choose the module that facilitates the modeling of walls and solid slabs.

-  Analysis of a shell.

The main window will appear (Fig II.2), which contains the default menu and toolbars at the top and bottom, and on the right, and the Object Manager window on the left. We will see later how to customize the default workspace and the toolbars.

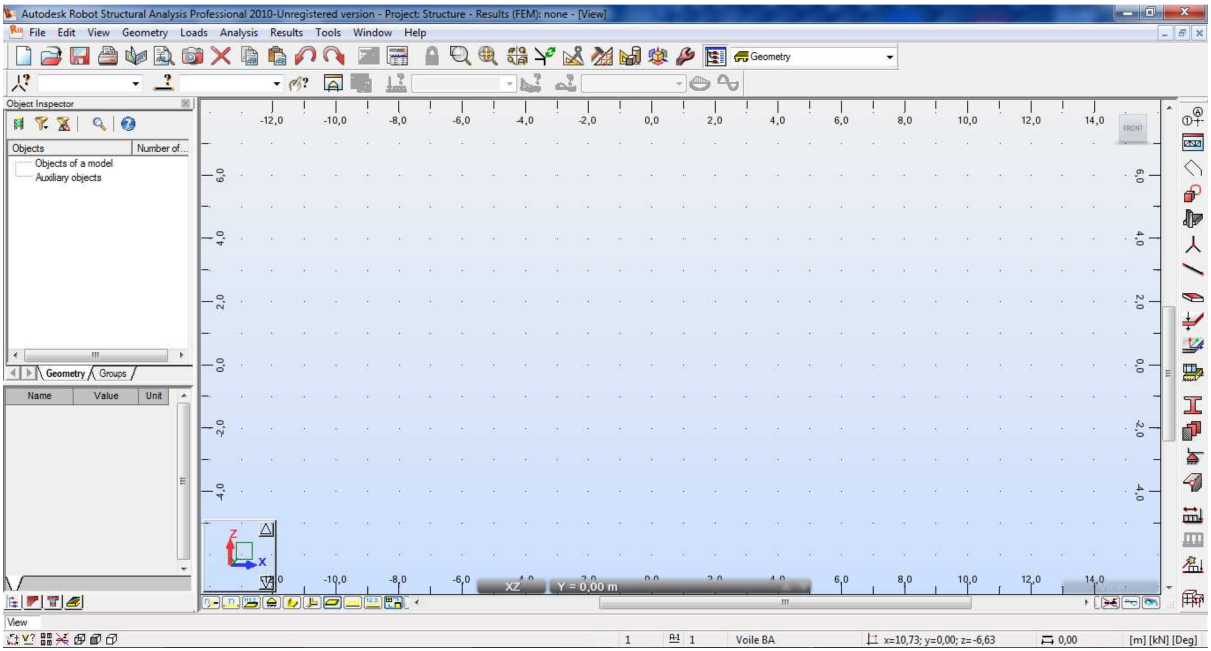


Figure II.2: The main windows of the software.

II.2 Adjusting preferences and project-specific preferences

To adjust preferences (language, display, etc.) and project-specific preferences (units and formats, materials, catalogs, design standards, etc.), click on the menu (Fig II.3):

Figure II.3: General preferences of the software.

Through this window, you can, for example, change the working language from French to English.

You can change the background color by clicking on View (Fig II.4).

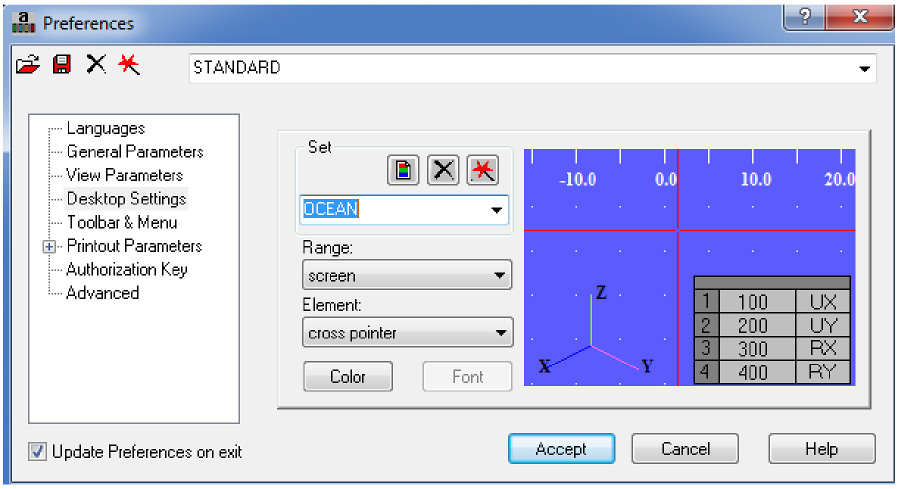


Figure II.4: Changing the color of the graphical interface.

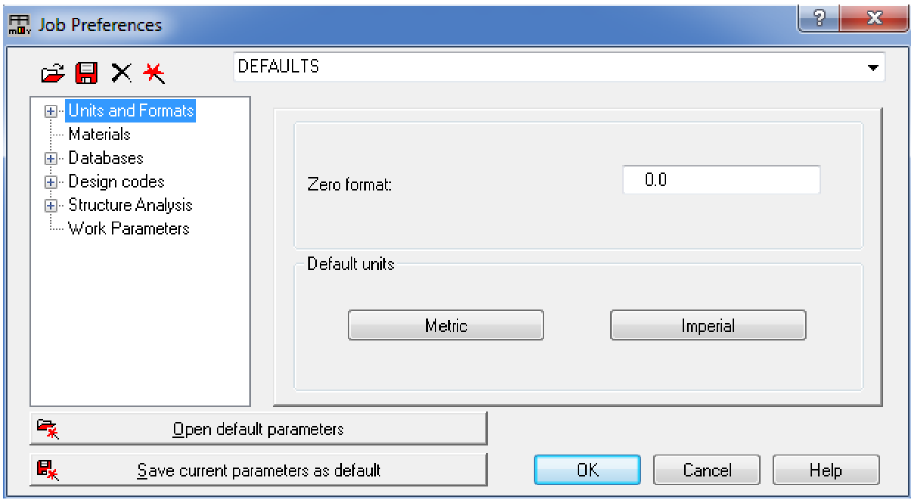


Figure II.5: Job preferences of the software.

II.2.1 Units and formats

Through this window, you can modify the units for dimensions, forces, angles, and displacements, among others (Fig II.6).

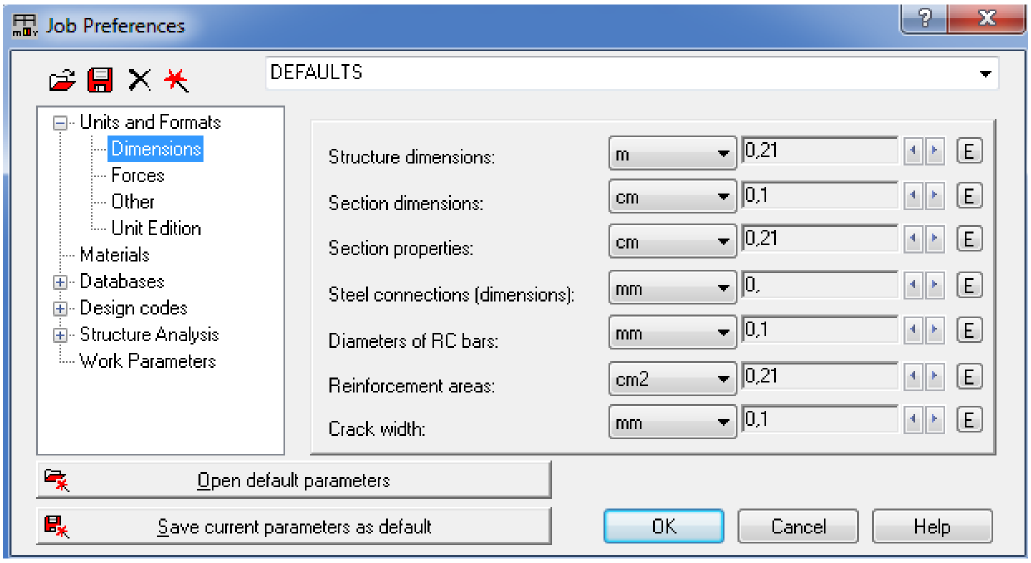


Figure II.6: Adjusting job preferences.

You can also change the number of decimal places by clicking on the arrows (Fig II.7):

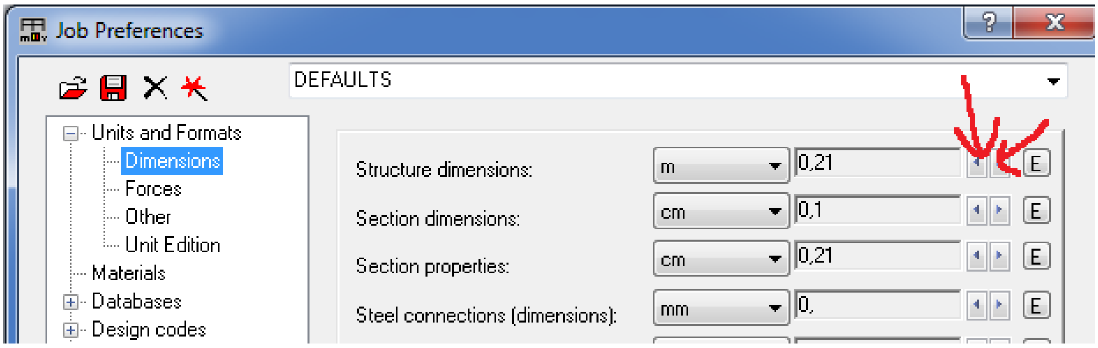


Figure II.7: Changing the number of decimals.

II.2.2 Design standards

Robot contains various regulations and standards, and you can select the standard used in your country from the dropdown menu (Fig II.8):

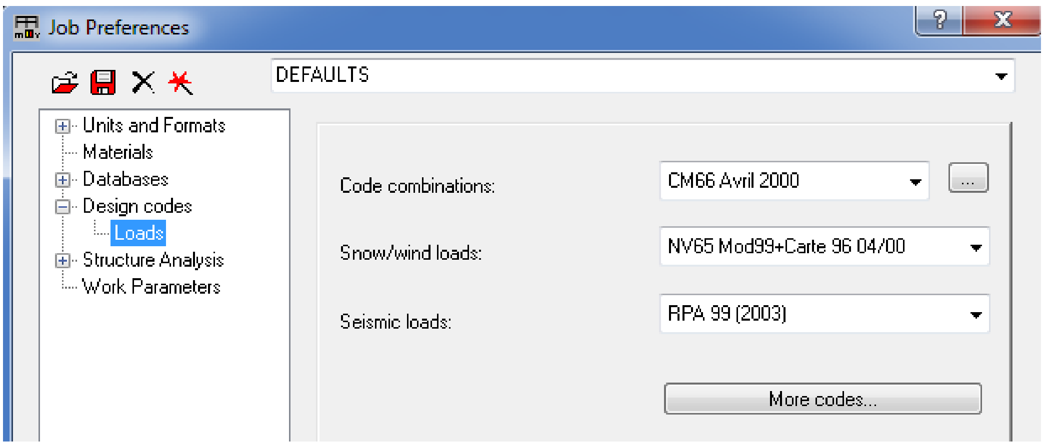


Figure II.8: Various regulations and standards of the software.

The same applies to seismic and climatic loads (Fig II.9):

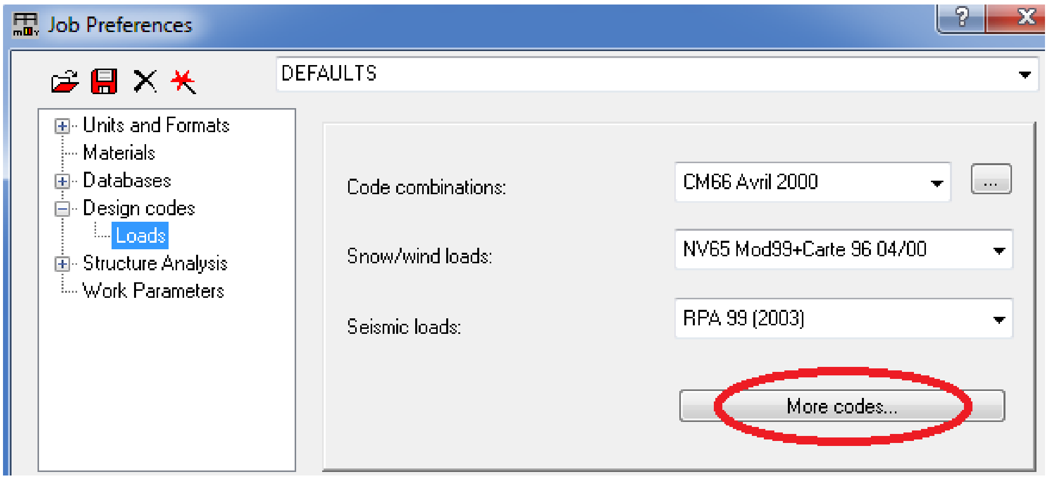


Figure II.9: Seismic and climatic loads available in the software.

If the standard you are looking for is not in the dropdown menu, you can add it from the list of standards in the menu by clicking on "More Standards" (Fig II.10):

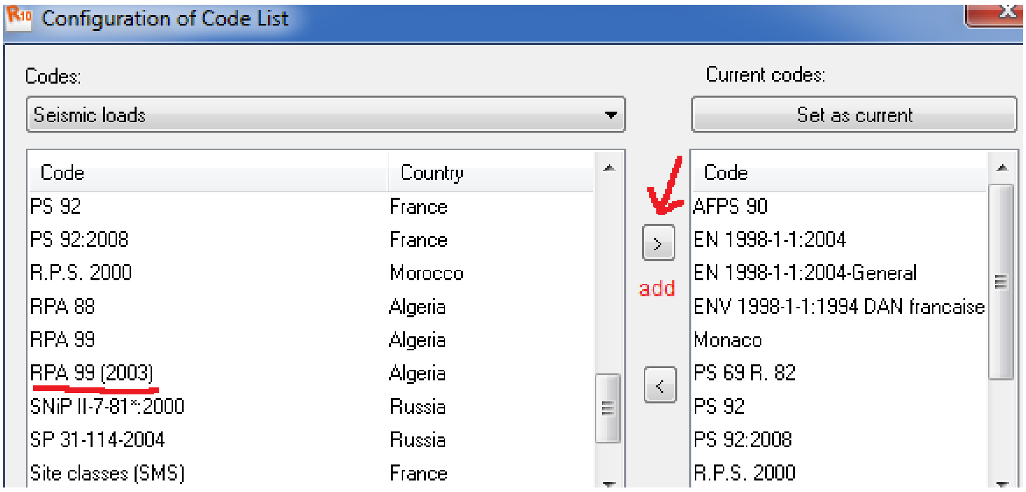


Figure II.10: Adding more standards in the software.

<u>**Note:**</u> Preference adjustment is done only once when starting the project. If you have multiple project types, each with its own preferences (units, standards, etc.), with Robot, you can define multiple preferences and save each preference in a file. If you want to use a specific preference, you only need to open the corresponding file for the desired preference.

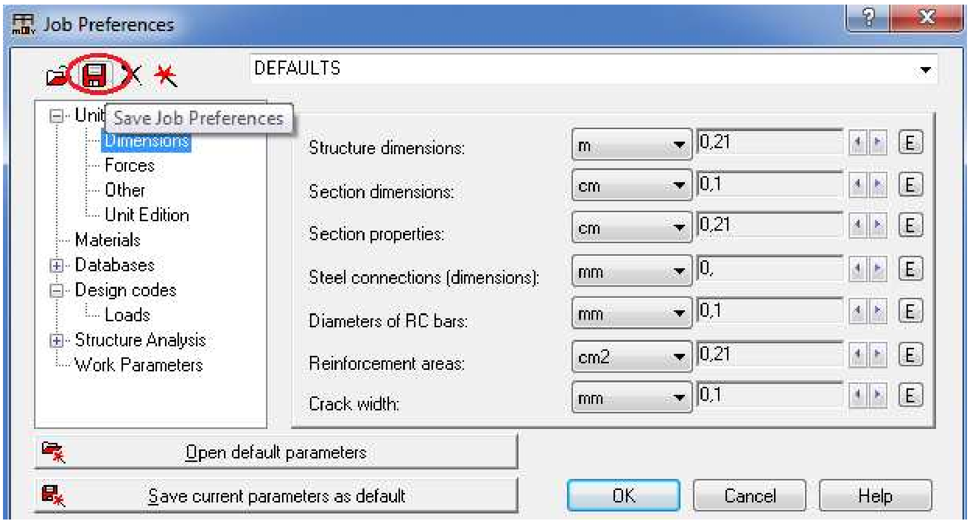


Figure II.11: Saving the applied modifications.

II.3 Modeling of Structural using RSA Software

The representation of a real structure, whether it's made of concrete or steel, through a numerical model using Robot 2010 software requires:

- Defining the construction lines of the structure in all three directions.
- Defining the sections of the elements that make up the structure (beam elements or panels).
- Graphically representing and drawing the structure using the defined elements.
- Defining the supports within the structure.
- Defining load cases and combinations, and applying loads to the structure.

In the following, you will find the steps to follow for modeling a structure using Robot 2010. We have taken an example of a simple reinforced concrete portal frame structure for illustration. The structure is shown in the figure (Fig II.12):

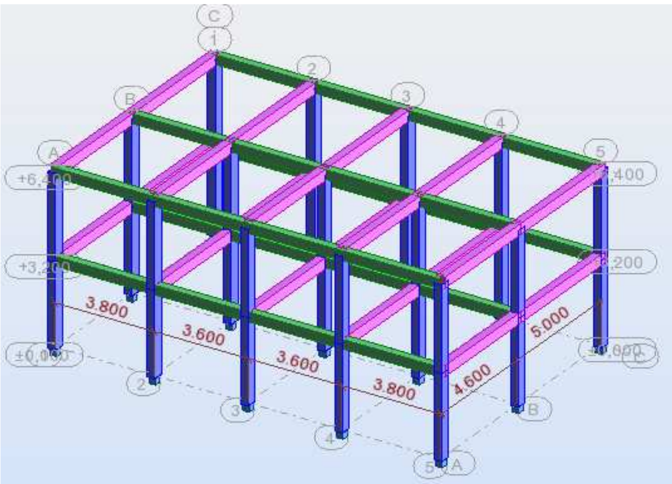


Figure II.12: 3D view of the structure.

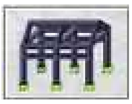
II.3.1 Construction lines

Construction lines or the grid of the structure represent the axes of the elements of the structure to be modeled in the three directions X, Y, Z, as well as the endpoints of the elements and the edges of the structure's faces.

On these lines and their intersections, you can subsequently draw the bars, beams, and other components, connecting them easily. For this purpose, you need the dimensions of the structure (length, width, height), along with detailed information about the spacing and dimensions of the structural elements, essentially a detailed structural plan.

Our example is a reinforced concrete structure consisting of 5 identical parallel portal frames connected by beams, with the following dimensions:

- Structure height = 6.40 m
- Structure length = 14.80 m
- Structure width = 9.60 m

At the start of the Robot 2010 software, we select the module.  **Analysis of a spatial portal frame.** The main window appears, and we begin drawing the construction lines using the command.  the first icon on the toolbar located on the right side of the window (Fig II.13).

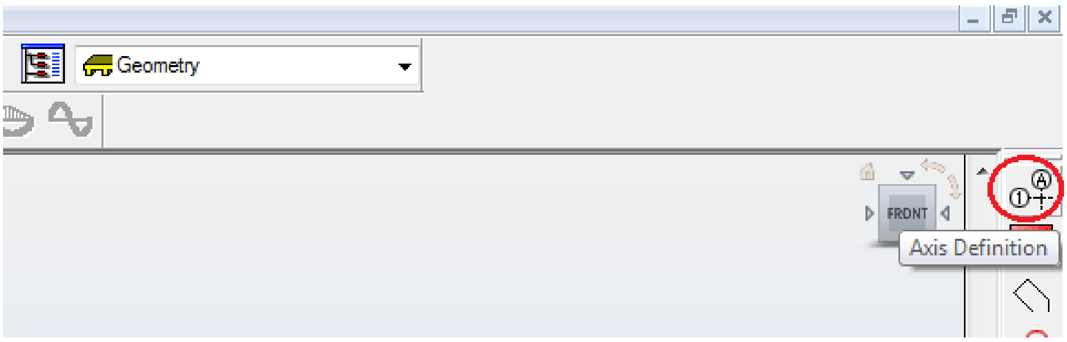


Figure II.13: Icon of the construction lines.

By clicking on this icon, the following dialog box opens (Fig II.14):

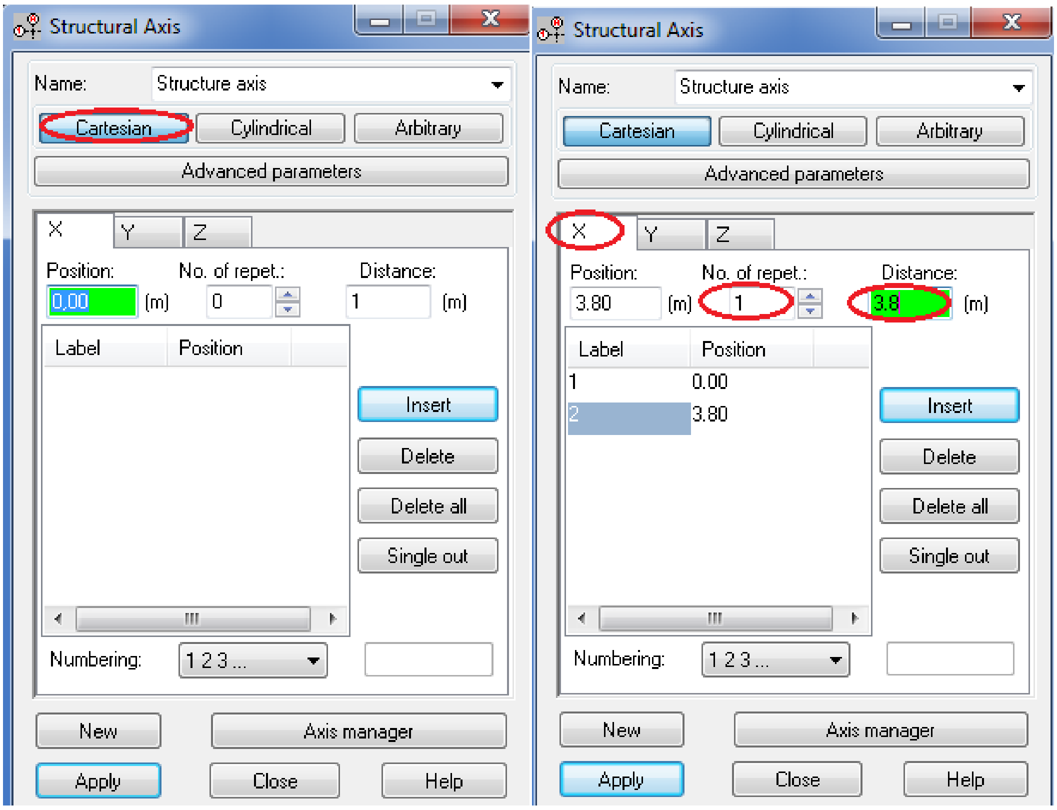


Figure II.14: Drawing of the construction lines.

We use Cartesian coordinates X, Y, Z. In the "Position" field, enter the value of the distance from the reference axis 0m that you want to draw from.

In the "**Repeat x**" and "**Spacing**" fields, leave 0 and 1m if you don't have a consistent spacing between elements. For instance, in our case, in the Z direction, there's a spacing of 3.2m between two levels. So, in the "Repeat x" field, put 2, and in the "Spacing" field, put 3.2.

We perform this operation for all three axes (X, Y, and Z).

For our example, we need to input the following series of values:

For X axis: 0, 3.8, 7.4, 11.0, 14.8 m
For Y axis: 0, 4.6, 9.6 m
For Z axis: 0, 3.2, 6.4 m

Click on **Apply** to save:

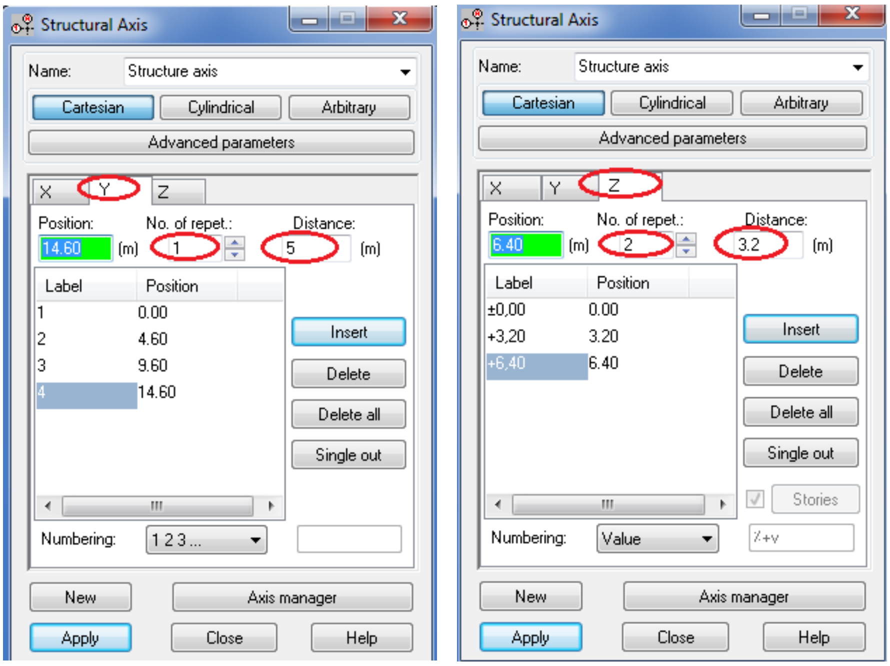


Figure II.15: Saving the construction lines.

Note:

You can add construction lines at any time by inserting values in the desired direction, and you can also remove existing lines by selecting the number and clicking on **Delete.**

You can also make lines **bold.**

In the same project, you can define multiple sets of construction lines using the **New** option in the construction lines dialog box.

You can also manage these lines (delete, activate, or deactivate specific lines) using the **Lines Manager** option.

When you activate the 3D view, you will get the following result (Fig II.16):

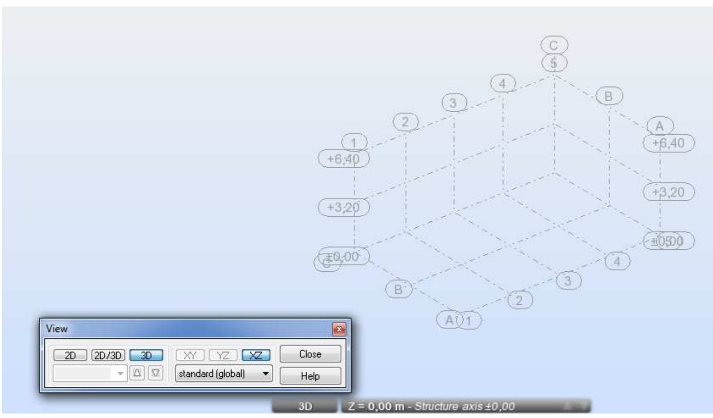


Figure II.16: 3D view of the construction lines.

II.3.2 Section definitions

To define the sections of beam elements, we use the command  "**Beam Profiles.**"

With this option, you can define the sections for all beam elements in the structure: columns, beams, whether they are made of concrete, steel, wood, etc (Fig II.17):

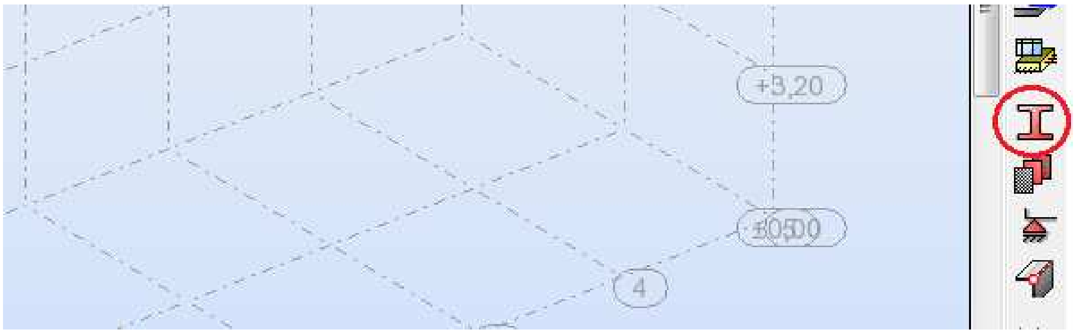
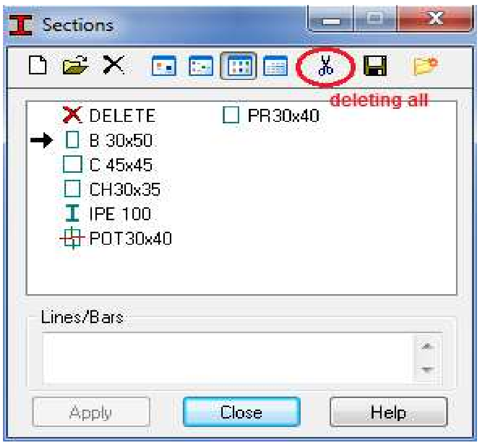


Figure II.17: Definition of the sections.

By clicking on the icon, the following dialog box opens, and by using the "Delete all unused sections" option, you can remove the default sections provided by the software (Fig II.18):



You will get:

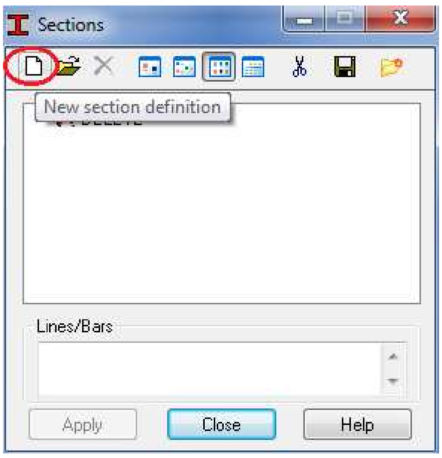


Figure II.18: Removing the default sections provided by the software.

Click on **New** to define the desired sections, and the following dialog box will open (Fig II.19):

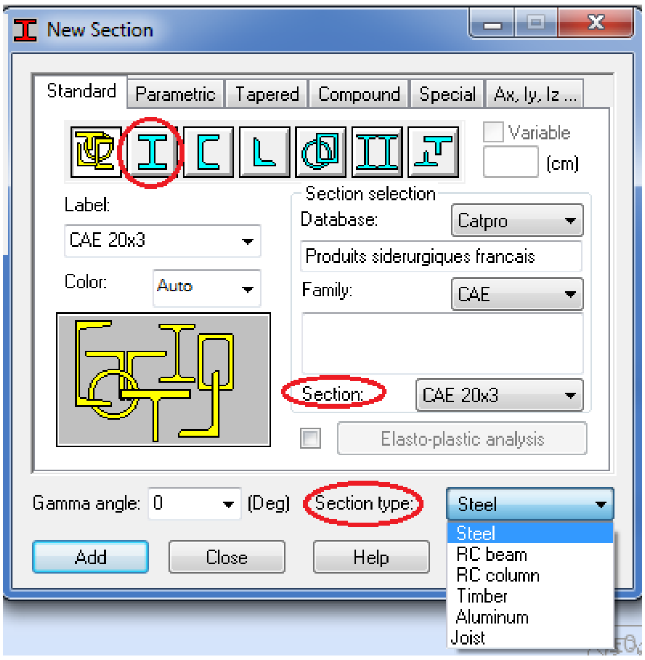


Figure II.19: Selection type of the section.

Select the **Profile Type** according to the case, whether it's steel, concrete beam, concrete column, etc.

In our example, since it's a reinforced concrete structure, select "**Concrete Column**." We are going to define a section (30x40) for the columns, a section (30x40) for the main beams, and a section (30x35) for the secondary beams. To do this, click on the "I-Section" (encircled icon in the image).

In the general menu, select the column's section type, for example, "**Rectangular**," and in the "**Dimensions**" section, input the desired dimensions, for example, b = 30 and h = 40. Assign a name to the section, for instance, "**COL(30x40)**," and for the gamma angle, set gamma = 90°.

Click on "**Add**" to save your selection in the profile list.

We do the same thing for the other sections, and we'll have (Fig II.20):

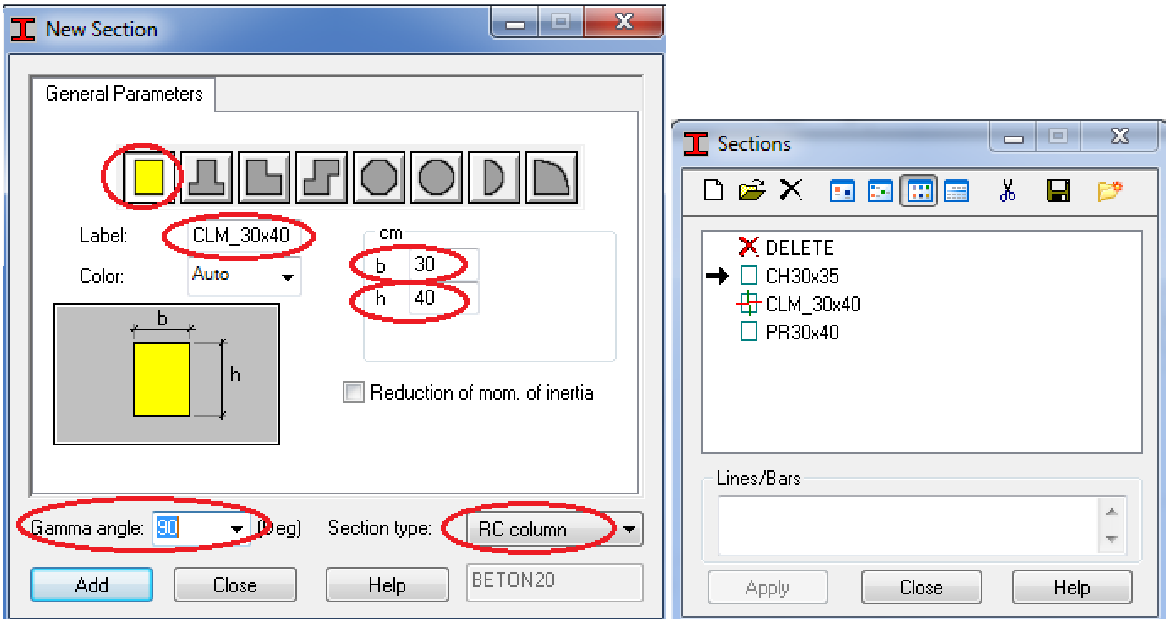


Figure II.20: Creating the column and beam sections.

You can give the new section any name you want and choose the profile's color; otherwise, the software will use default settings.

II.3.3 Structure Definition

Now that we have defined the construction lines and the sections of the structural elements, we can begin drawing our structure using the previously created construction lines.

We enable the **XYZ 3D view**, which provides a three-dimensional perspective of the structure. To prevent modeling errors, we disable grid snapping. To do this, click on the snap mode icon located at the bottom left corner of the window.

The **Snap Mode** dialog box opens, uncheck the "Grid" box to disable grid snapping (Fig II.21):

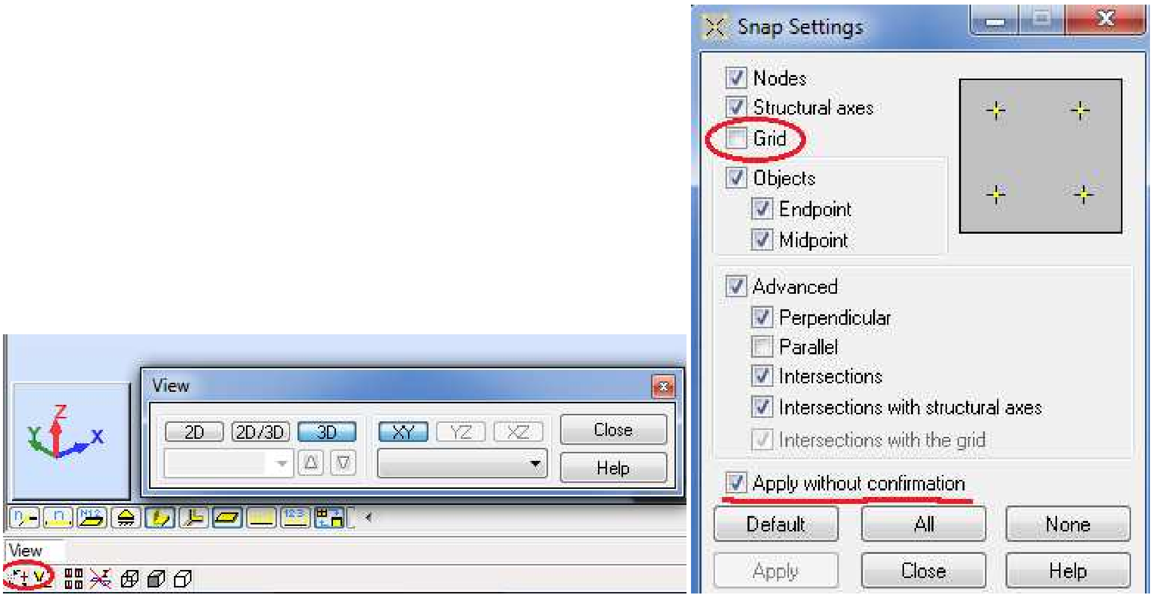


Figure II.21: Snap setting dialog box.

Click on the command.  **Bars**

The following dialog box appears (Fig II.22). In the "**Type**" field, select "**Column**," and in the "**Section**" field, select "COL(30x40)." Now, click on the "**Origin**" field, and begin drawing the 3 columns of this side using the construction lines. Next, move on to drawing the beams using the same principle: select "Beam" as the type and "BEAM(30x40)" as the section.

So we have defined the first portal of our structure (Fig II.23).

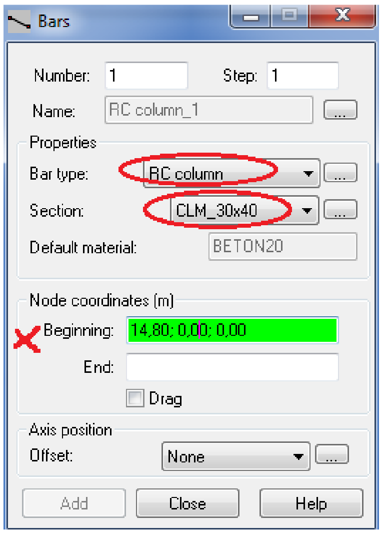


Figure II.22: Creation of the bar elements.

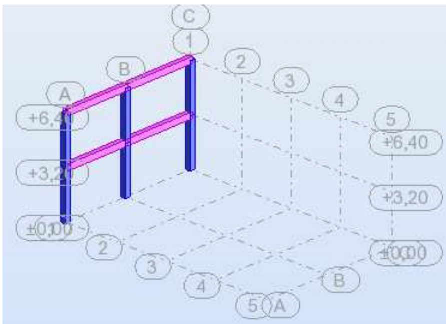
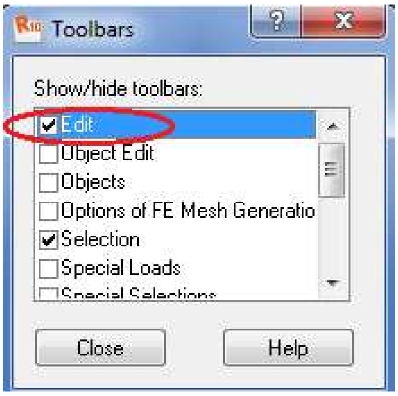


Figure II.23: Construction of the first portal of the structure.

Note:

You can always customize your working environment by displaying toolbars. For example, if you want to show the "Edit" toolbar that includes the "Split Beams" option, you can go to "**Tools**" > "**Customize**" > "**Show Toolbars**."

The following dialog box will open:



By checking the "Edit" checkbox, you will have the "Edit" toolbar displayed (Fig II.24).

Figure II.24: Selection and displaying the edit toolbar.

Select all of the portal frame except for the nodes at the base of the columns. To do this, click and drag from the right to the left, as shown in the figure (Fig II.25).

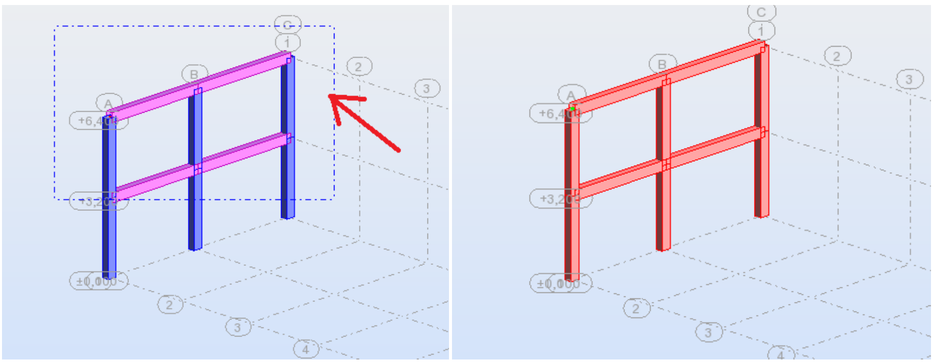
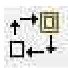


Figure II.25: Selection of the portal.

After the selection, click on  "Translation." The following dialog boxes will open:

In the "Number of repetitions" field, enter the number of portal frames you want to create. In the "Translation vector" field, input the distance between the portal frames (dx, dy, dz) = 3.8, 0, 0 m, and then click on "Apply" (Fig II.26).

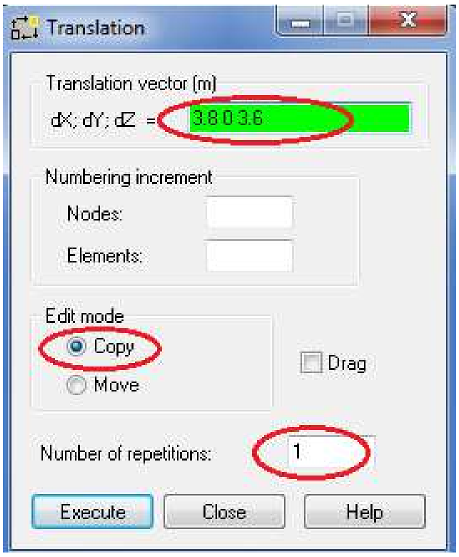


Figure II.26: Setting of the translation dialog box.

To hide the construction lines, right-click and then click on "Attributes." In the opened dialog box, click on "Structure" and uncheck the construction lines option (Fig II.27).

To get a clearer view of the drawn structure, click on "Profiles Sketch."

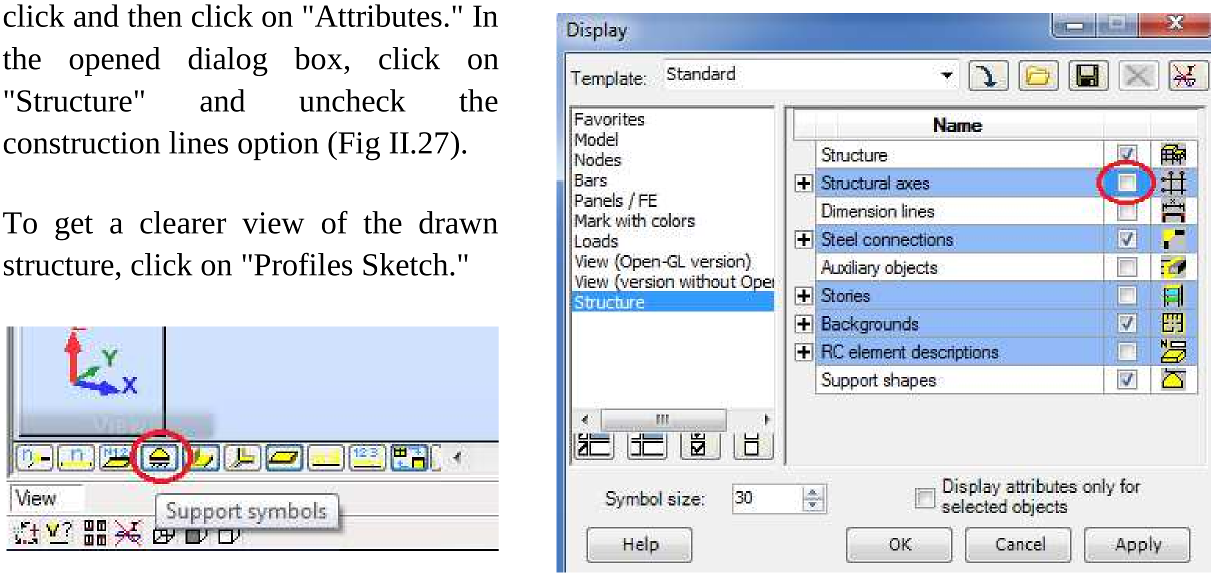


Figure II.27: Hiding the construction lines of the model.

You will have the following 3D view:

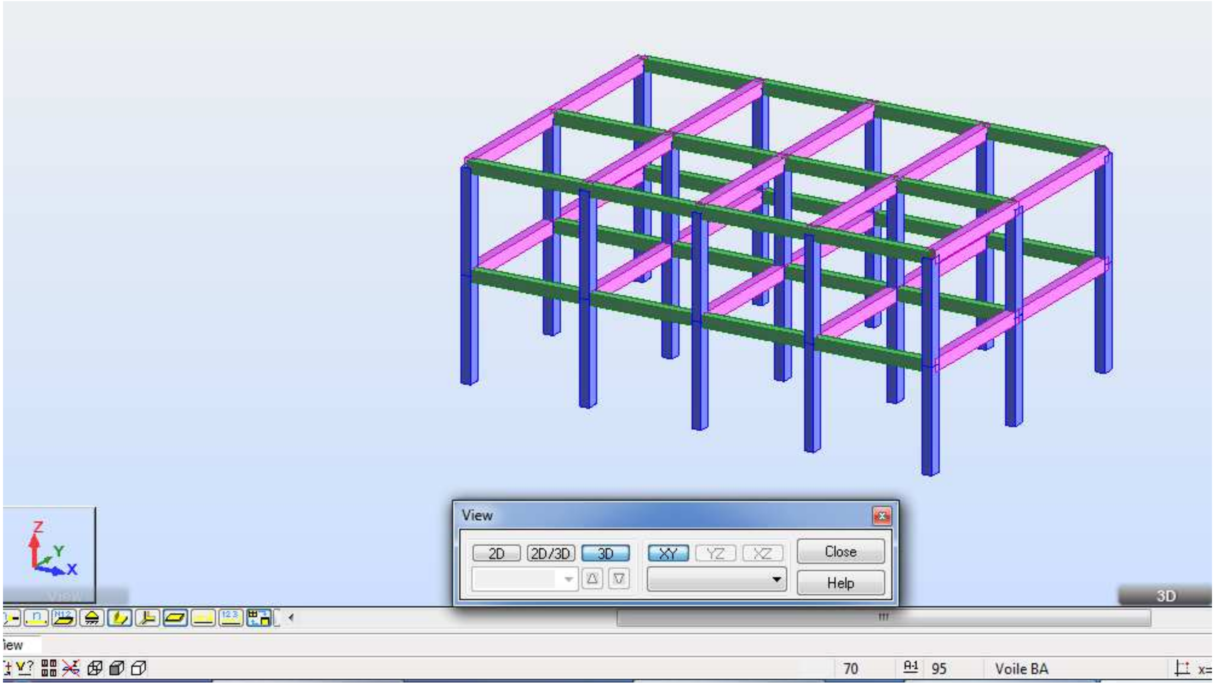
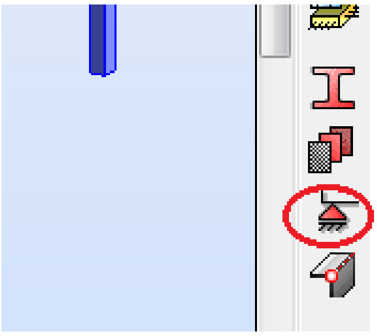
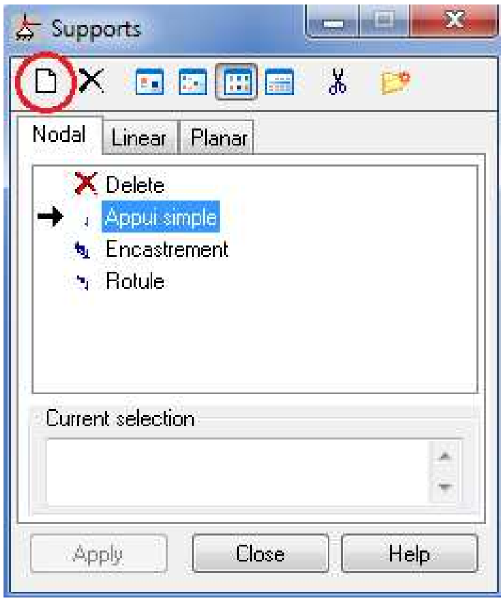


Figure II.28: 3D view of the model.

II.3.4 Support conditions

To define nodal supports within a structure, the "Supports"  command is used.





You can choose the support type directly through this dialog box, or you can define a new support using the "Define New Support" option. By clicking on it, the following dialog box will open (Fig II.28):

Figure II.28: Supports dialog box.

Through the "Support Definition" dialog box, you can define the directions to be restricted by checking the displacement and rotation boxes along the axes, as illustrated in the figure. These could be linear displacements (UX, UY, UZ) or angular rotations (RX, RY, RZ).

For instance, for a fixed support, all displacements and rotations in all directions are blocked. For a hinge support, linear displacements (UX, UY, UZ) are blocked, and rotations (RX, RY, RZ) are free.

For our example, we will choose to apply the Fixed support type for the entire structure (Fig II.29).

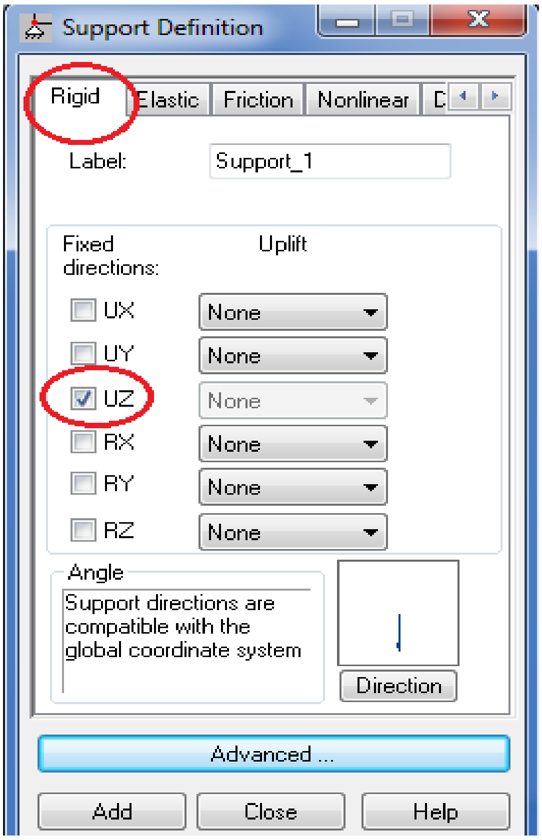


Figure II.29: Supports setting of the construction.

For that purpose, select "Fixed" in the Supports dialog box. Then, click in the "Current selection" field, and proceed to select all the nodes at the base of the columns. Finally, click on "Apply" (Fig II.30):

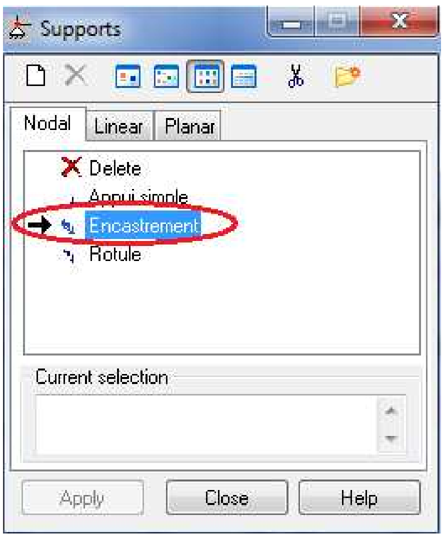


Figure II.30: Assignment of the supports.

By enabling the "Realistic Display" of the structure and showing the sketches of profiles, you will get the following 3D view (Fig II.31):

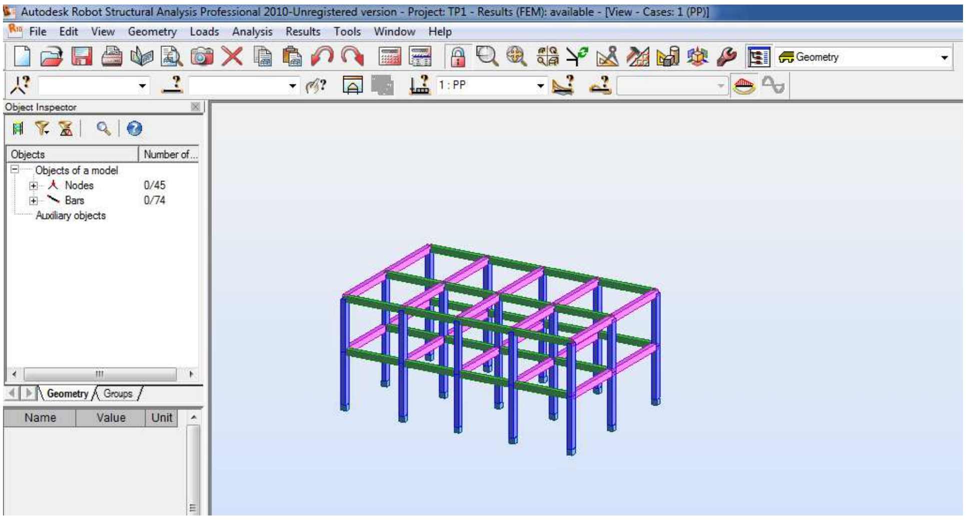


Figure II.31: 3D view of the model with supports.

II.3.5 Loading

Loading a structure involves defining load cases based on the nature of the loads (permanent, live, seismic, etc.), then applying the loads (loads on members, surface loads, etc.) to the structure for the created load cases. Finally, combinations of load cases are defined.

a- Load cases

To define load cases, click on the "Load Cases"  command.

Through this dialogue box, the Nature of the load case to be defined is selected, as illustrated in the figure.

In the "Name" field, a name can be assigned to the load case, or the default name suggested by the software can be used.

Subsequently, clicking "New" adds the load case to the list of defined cases (Fig II.32).

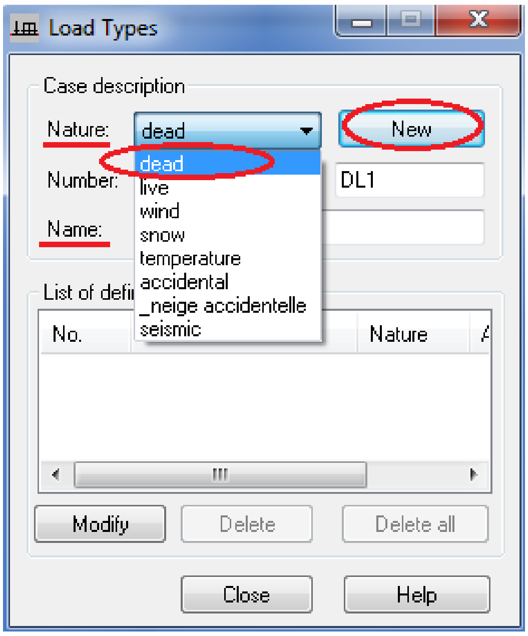
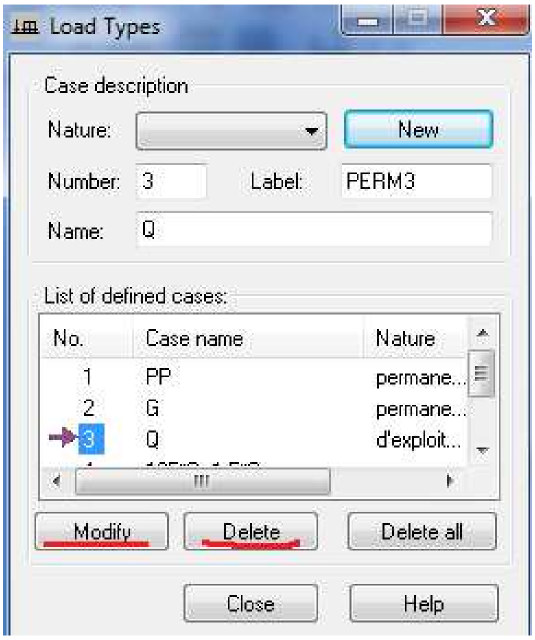


Figure II.32: Load dialog box.

For our example, in addition to the self-weight of the structure, we will define a dead load and a live load, both distributed over the rafters that connect the frames at the top.



We will name the self-weight of the entire structure as "pp" and then click on "New." The software automatically assigns the first defined permanent load case as the default self-weight of the structure.

For the other load, we will name it "G" for dead load and "Q" for live load. The load case's name, its position in the list, and its type can be modified, or it can be deleted using this dialog box (Fig II.33).

Upon clicking "Close," we proceed to apply the "G" and "Q" loads to our structure. We select the current load case using the top toolbar.

Figure II.33: Creation of the different load types.

b- Loads

For our example, we have three loads:

- Self-weight of the structure, PP, automatically calculated by the Robot 2010 software.
- A permanent load, G = 5 kN/m, distributed over the main beams.
- A permanent load, G = 2 kN/m, distributed over the secondary beams.
- A live load, Q = 2.5 kN/m, distributed over the main beams.
- A live load, Q = 1.0 kN/m, distributed over the secondary beams.

By selecting the PP case from the list illustrated in the previous figure, we will have (Fig II.34):

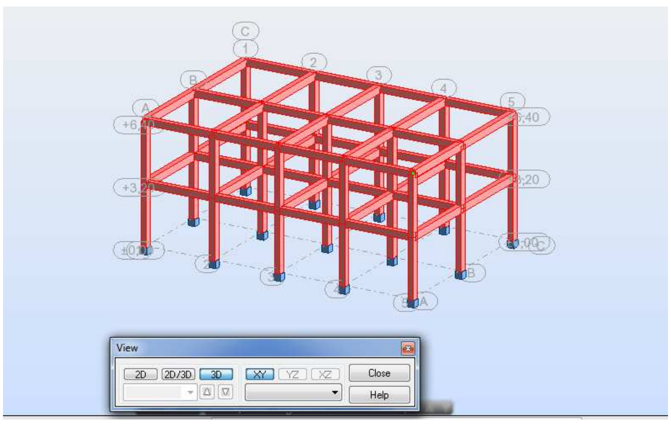


Figure II.34: Assigning the self-weight of the structure.

To define the permanent load G, we select the G case from the list of load cases and click on the "Define Loads" command: 

In the dialogue box that opens, we choose "Beams" and click on the symbol for "Uniform Load," as shown in (Fig II.35):

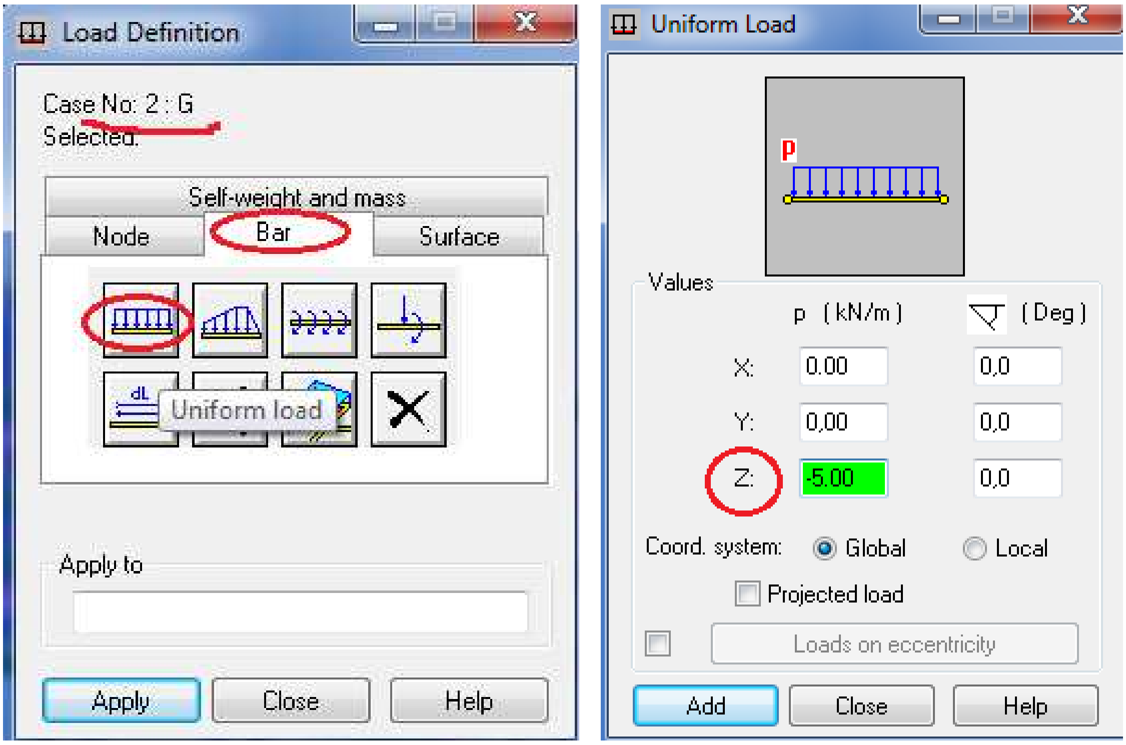


Figure II.35: Definition of the applied loads on beams.

We enter the value of the load in the Z direction (-5 kN/m) in the field labeled Z, and then click on "Add." Now, using the "Select Beams" tool,

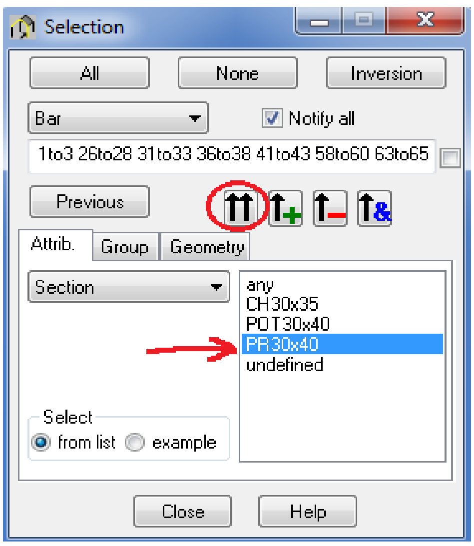


Figure II.36: Applying loads on beams.

we select all the bars with the PR30x40 profiles as shown in the figure and then click on "Close."

Returning to the Load dialogue box, in the field labeled "Apply to All," all PR30x40 bars are selected, so we click on "Apply" to apply the load to the beams (Fig II.36).

We repeat the same process for the live load Q.

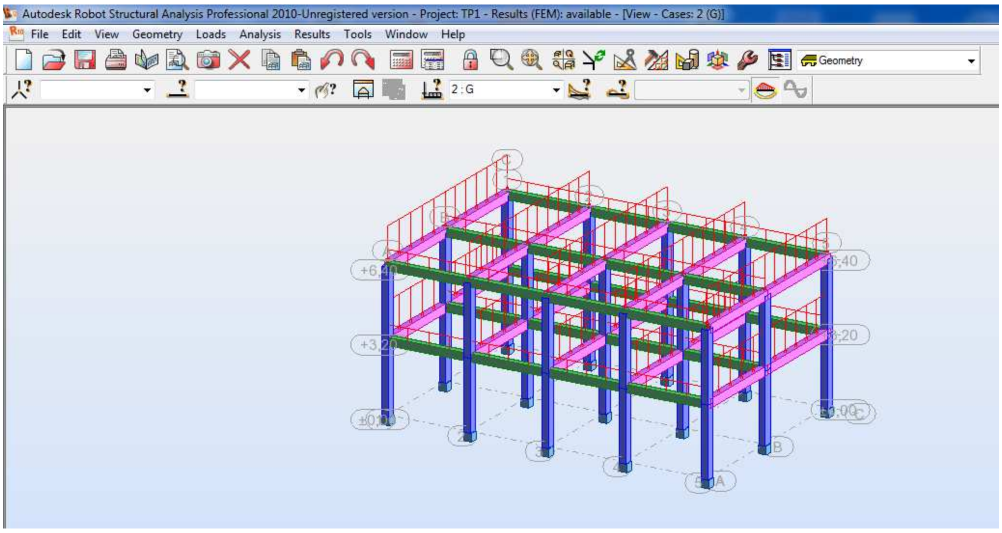


Figure II.37: Visualizing applied loads in the 3D view of the model.

II.3.6 Load case combinations

For our example, we will define the combination 1.35 G + 1.5 Q.

To define load case combinations, we use the "Manual Combinations" command located in the "Loads" menu ► "Manual Combinations." The following dialogue box opens:

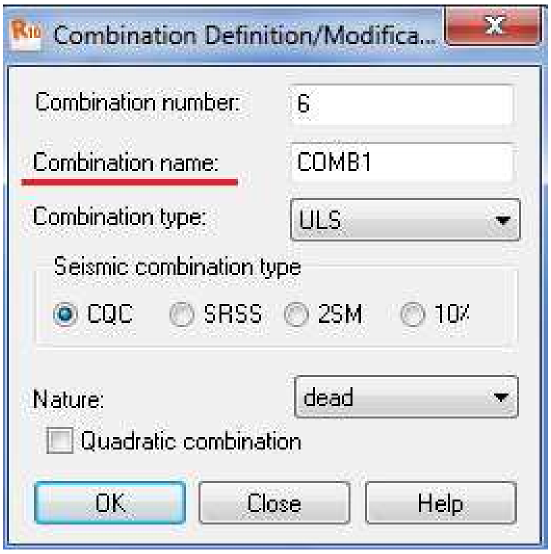


Figure II.38: Combinations dialog box.

We select the Combination Type and provide a name for the combination we are about to define.

For instance, we can name it "1.35 G + 1.5 Q," and then click on "OK."

In the dialogue box that appears, we will define our combination using the previously defined load cases (Fig II.38).

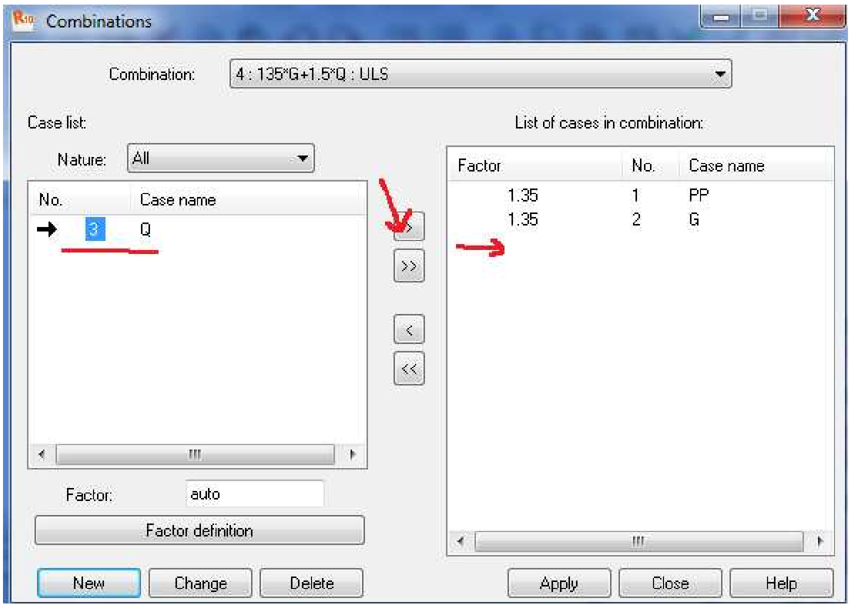
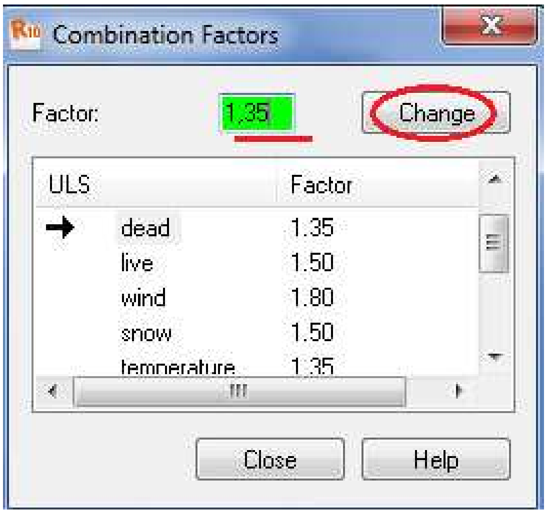


Figure II.39: Creation of the ULS combination.



If the coefficients you wish to apply are different from the automatic coefficients of the defined combination, you can set them by clicking on "Define Coefficients."

In the "Coefficient" field, you input the desired value and click "Modify" (Fig II.40).

After completing this step, you click "Apply" to save the combination.

You can also define another combination by clicking on "New."

Figure II.40: Definition and modifying the combination factors.

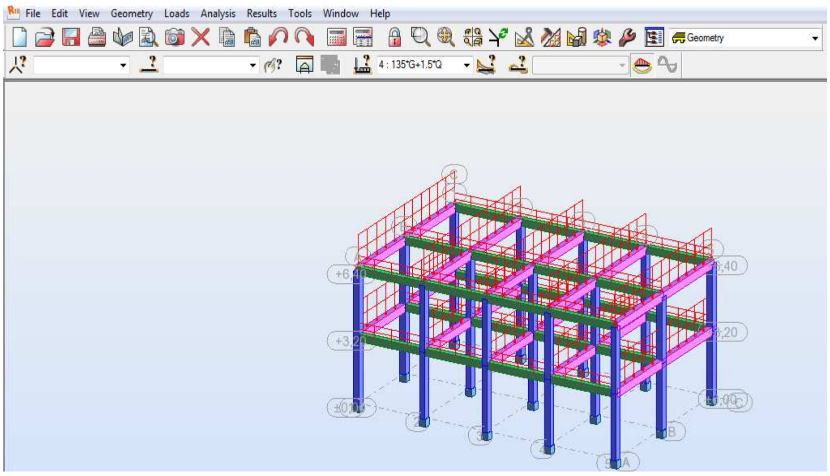


Figure II.41: Visualizing combination loads in the 3D view of the model.

II.3.6 Analysis of a structure

Now that we have completed the modeling of our example, we proceed to the calculation and analysis of this structure under the defined loadings.

Before running the calculation, it's important to verify the structure for modeling errors and disconnected bars. To do this, click on "Analysis" ► "Check Structure", (Fig II.42).

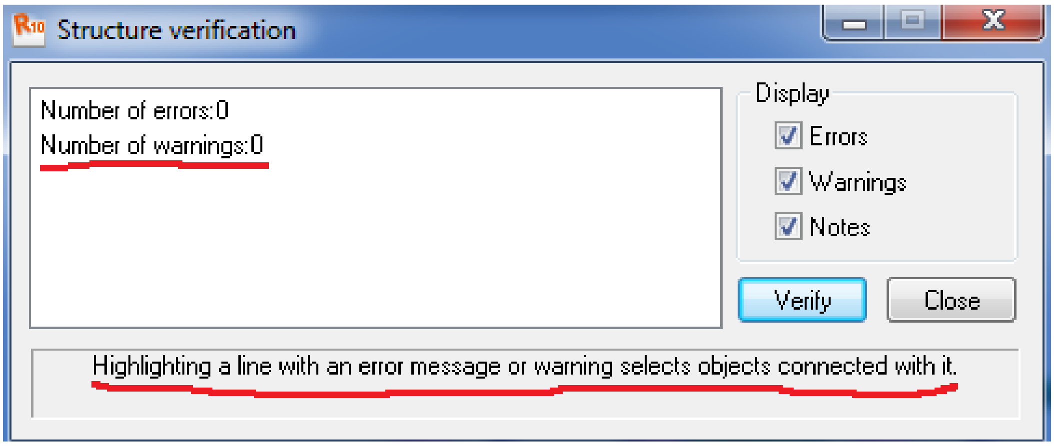
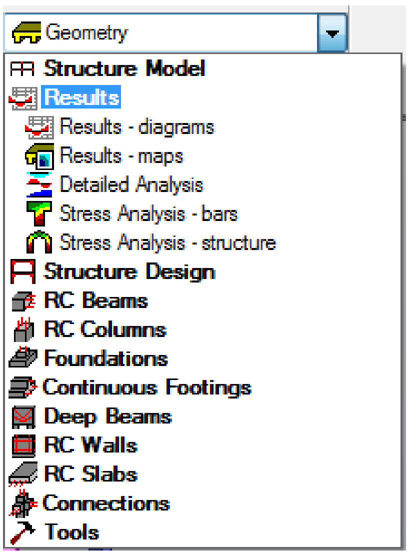


Figure II.42: Structure verification dialog box.

In the dialogue box, error messages indicate the specific errors and the corresponding objects causing those errors.

To initiate the calculation, click on the "Calculate" command  (Analysis ► Calculate).

II.3.7 Analysis results



To display the analysis results of the structure, including internal force diagrams, deformations, stresses, and reactions, select "Results" from the startup menu in the top toolbar.

You can also directly view result diagrams by going to "Results" ► "Bar Diagrams", (Fig II.43).

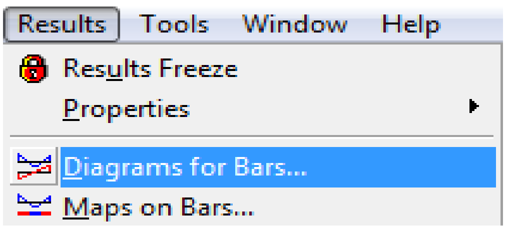


Figure II.43: Diagram for bars toolbar.

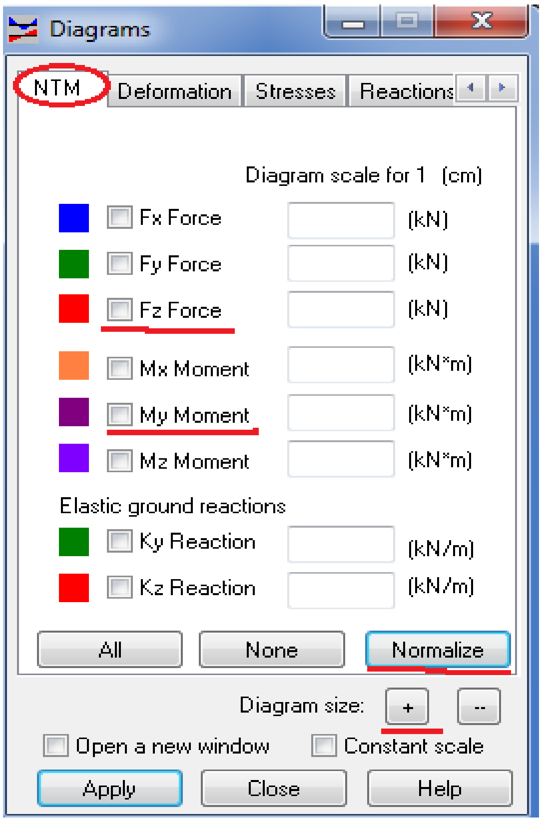


Figure II.44: Diagrams for bars dialog box.

The following dialog box opens:

To view the internal force diagram, you check the checkbox for the specific internal force, for example, the Moment My, and then click "Apply."

If the diagram's appearance is not satisfactory, you can click on "Normalize" to adjust the diagram's size.

You can also change the diagram's size by clicking on "+" and "-" buttons and then clicking "Apply", (Fig II.44).

Furthermore, you can modify the diagram settings to display the values of the forces directly on the diagram.

You can view the internal force and displacement diagrams of each individual element of the structure separately by right-clicking on the specific element and then clicking on "Object Properties", (Fig II.45).

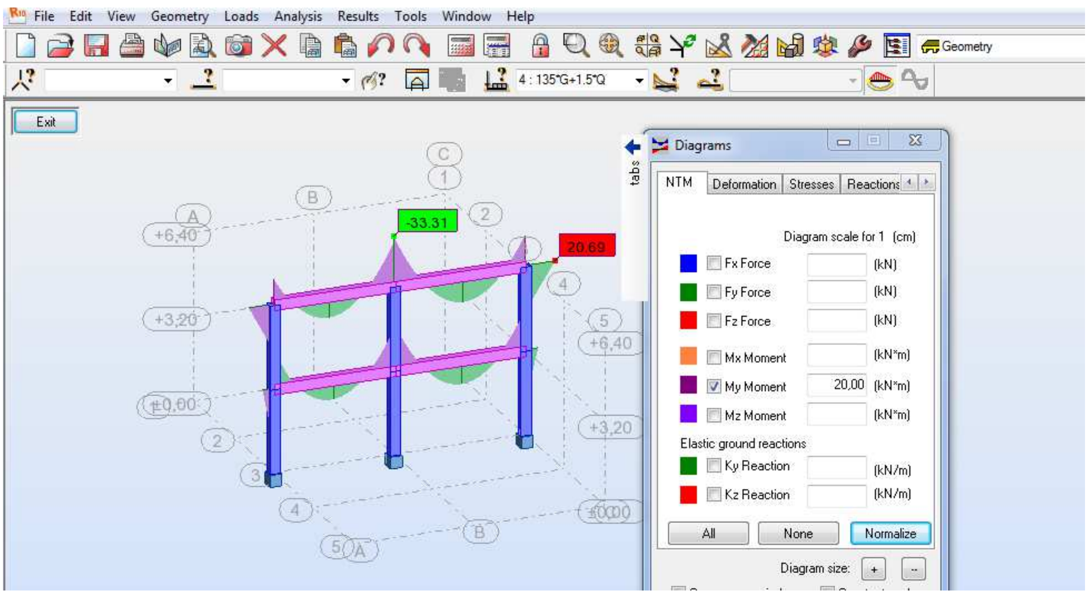


Figure II.45: Moment diagram of individual portal.

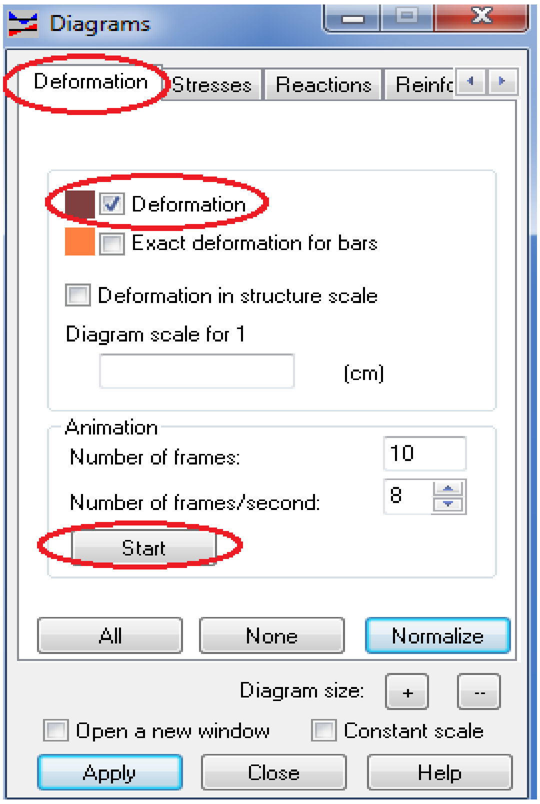
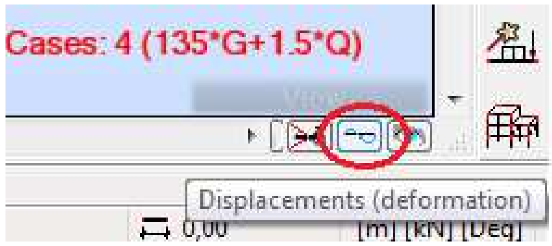


Figure II.46: Deformation dialog box.

To display the deformation of the structure, click on the "Displacements" icon located on the bottom right corner of the window (Fig II.46).



You can also display the deformation using the "Diagrams" dialogue box and start an animation by clicking on "Start" as shown in (Fig II.47).

Using the animation control bar, you can record this animation as a (.avi) file.

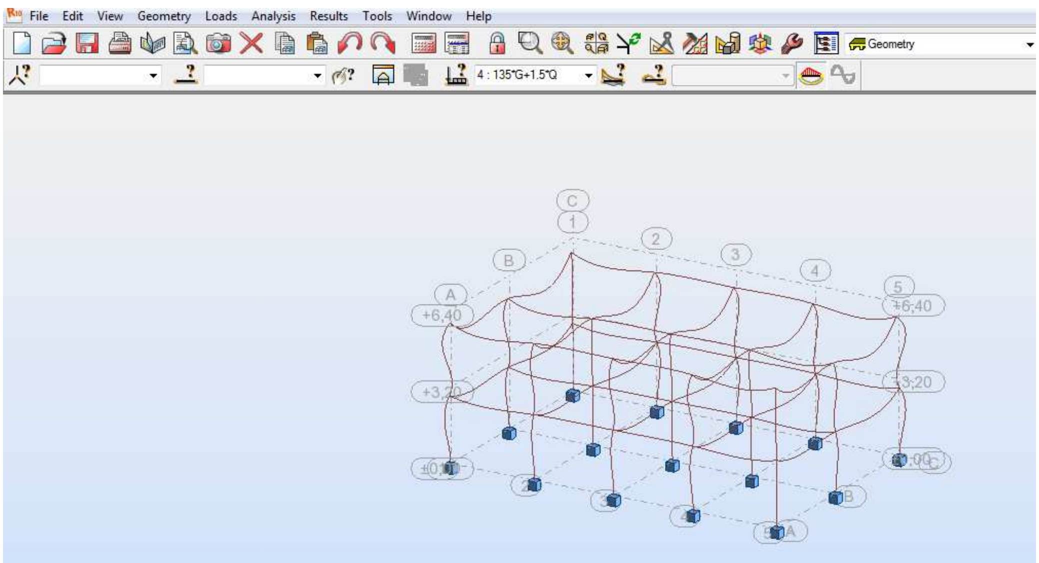


Figure II.47: 3D view of the structure deformation.

Chapter III

Study and monitoring of a real project using RSA 2010

III.1 Introduction

This application allows for the processing of the design of a simple reinforced concrete structure using the Robot software, in order to practice the software's usage for new users.

III.2 Presentation of the Structure

It is a residential building with 4 floors (ground floor + 4), located in the city of Guelma (zone IIa, according to RPA 99 version 2003), with a mixed structural system (shear walls + reinforced concrete frames) providing lateral resistance.

Number of floors: Ground floor + 4

Floor height: 3.20 m for all levels.

III.2.1 Dimensions of the structure

Building length = 21.4 m.

Building width = 10.75 m.

Total height = 16.0 m.

III.2.2 Dimensions of structural elements

Columns: 30x40 for all levels.

Beams: Main beams: 30x40 - Secondary beams: 30x35

Floor: Hollow core floor: 16+4

Solid slab: Solid slab with a thickness of 14 cm.

Staircase: Thickness of the landing = 16 cm

III.2.3 Load assessment

Typical floor: G = 5.0 kN/m²; Q = 1.5 kN/m² (bedrooms); Q = 3.5 kN/m² (balconies);

Roof floor (inaccessible): G = 6.3 kN/m²; Q = 1.0 kN/m² (inaccessible terrace).

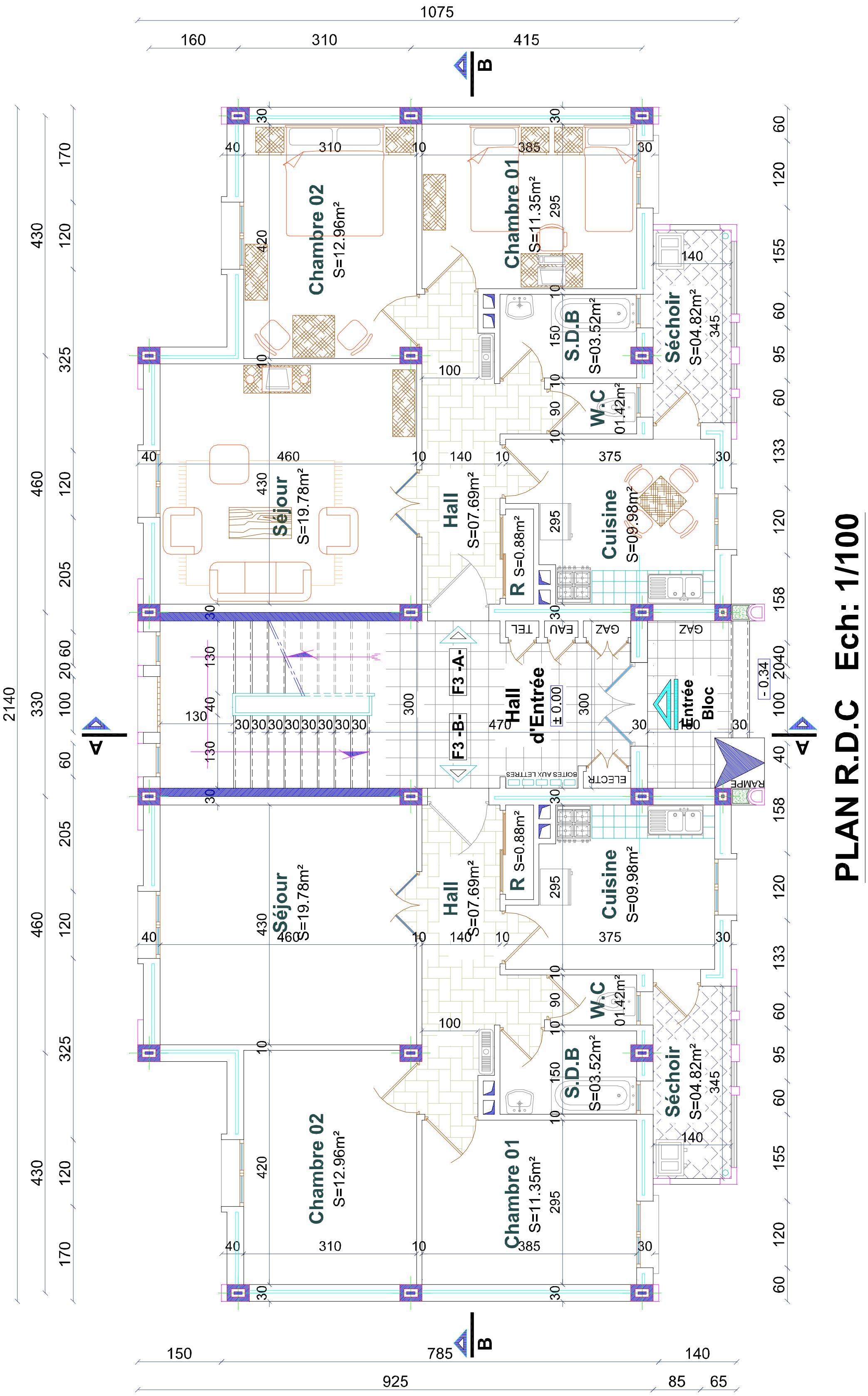


Figure III.1: Plan view of the studied structure.

III.3 Modeling

III.3.1 Project setup

At the start of the software, click on the "Shell Study" module, (Fig III.2), (the use of this module facilitates the modeling of walls and solid slabs):

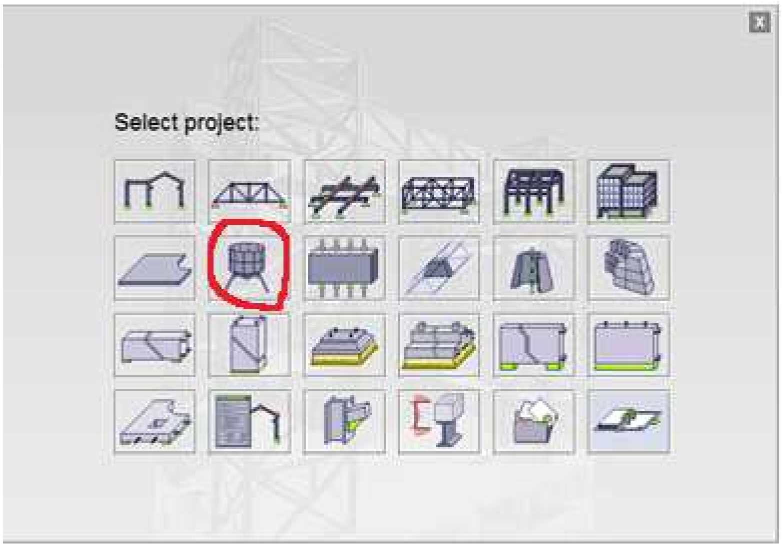


Figure III.2: Selection of shell study module.

III.3.2 Preference settings

Before starting the modeling, you need to adjust the preferences (language, display, etc.) and project preferences (Units, Materials, Standards, etc.). To do this, click on the dropdown menu Tools/Preferences (or Tools/Project Preferences), (Fig III.3).

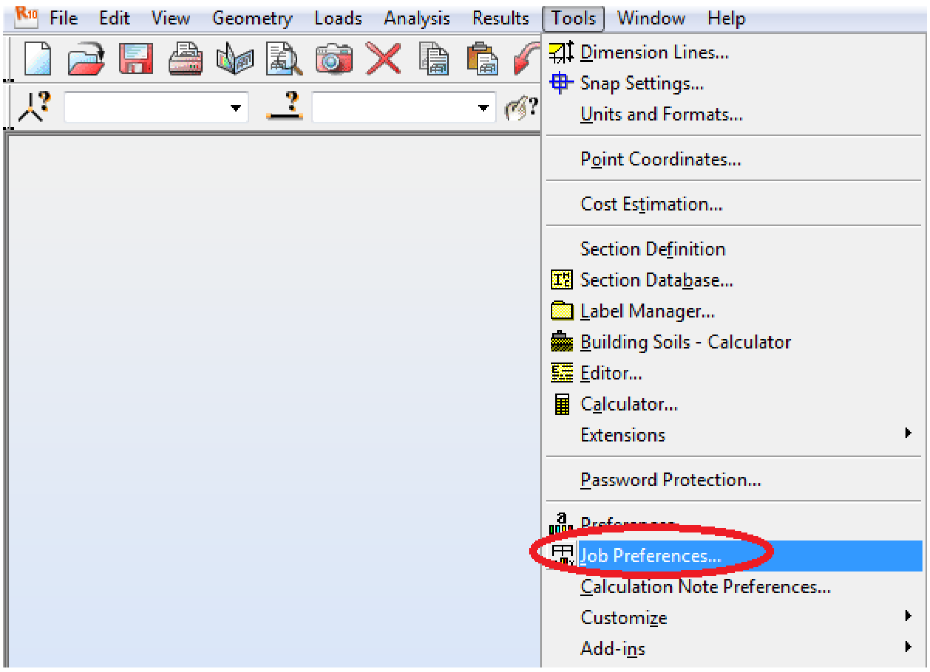


Figure III.3: Selection of job preferences.

Note: This procedure is done only once when you install the software.

III.3.3 Construction lines

The first step in modeling is drawing construction lines. These lines represent the axes of the structure (X, Y, and Z). In the Robot window, go to the first icon on the toolbar located on the right side of the window, (Fig III.4).

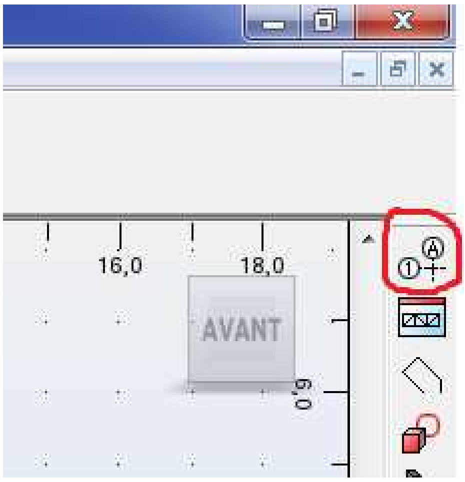


Figure III.4: Icon of the construction lines.

The following dialog box opens:

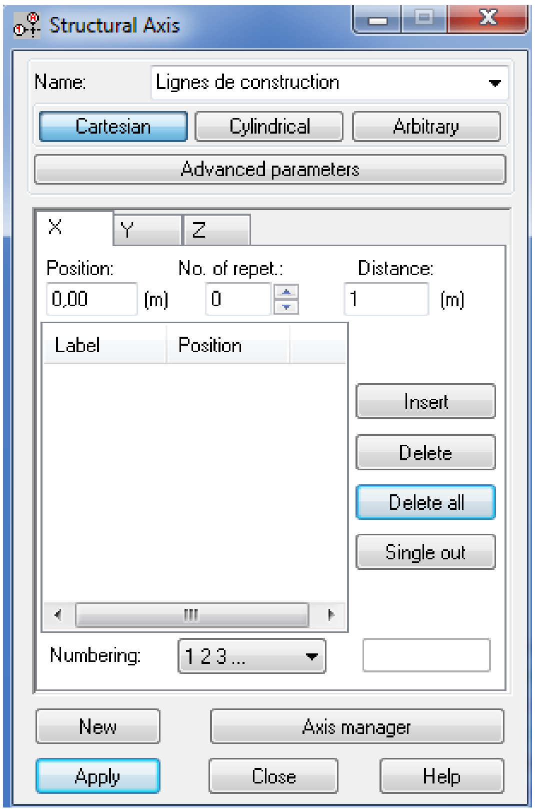


Figure III.5: Dialog box of the construction lines.

In the field (repeat), we must always enter the value 1 since we do not have repeating interaxial values (except for the Z axis where we can repeat 4 times 3.20).

In the field (spacing), enter the value of the pitch and each time click on (insert). We perform this operation for the three axes (X, Y, and Z).

The result should be as in Fig III.6):

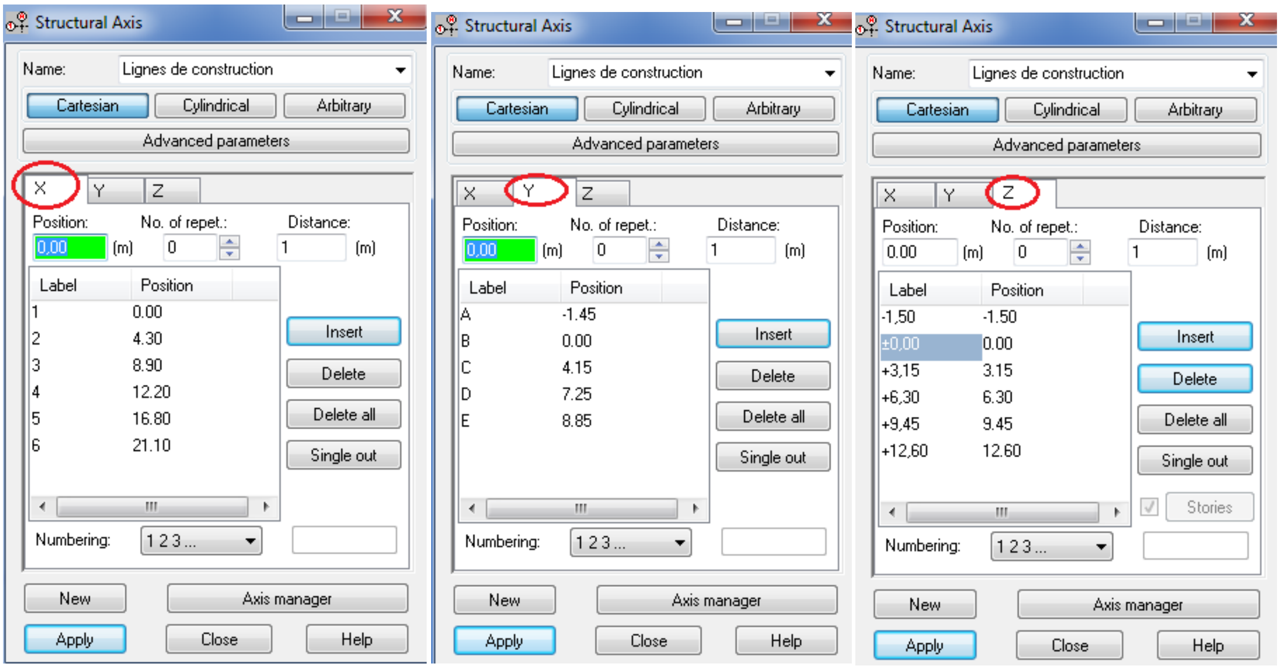


Figure III.6: Drawing of the construction lines.

Click on (apply) and activate the 3D view, we will have the following result, (Fig III.7):

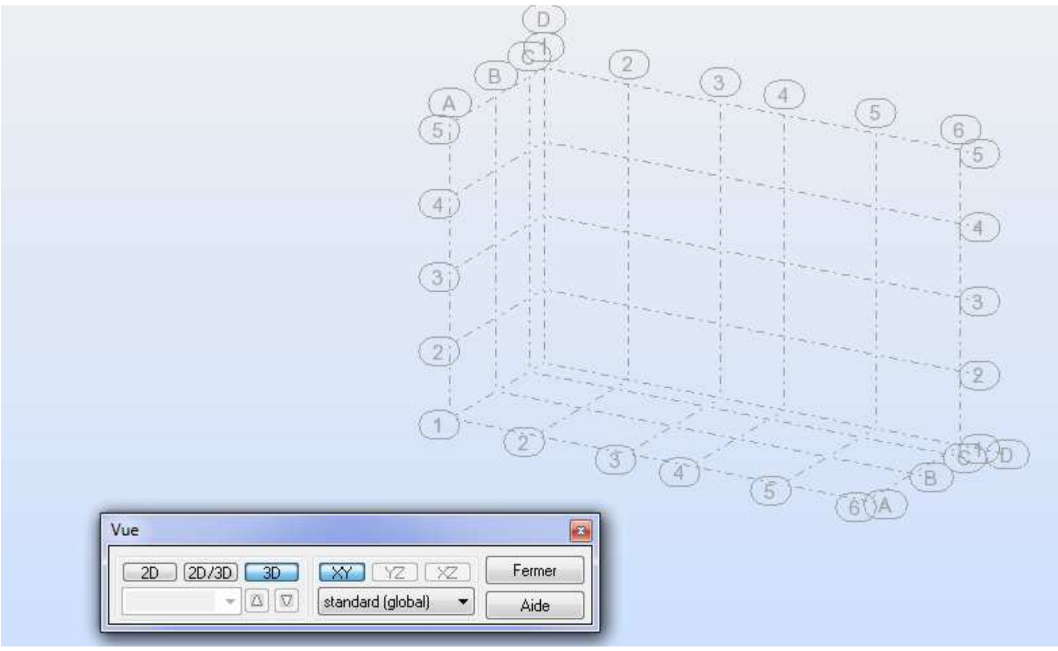


Figure III.7: 3D view of the construction lines.

Note:

- Multiple construction lines can be defined in the same project using the "new" option in the construction lines dialogue box. The management of these lines (deleting, activating, or deactivating desired lines) can also be done using the "line manager" option in the construction lines dialogue box, (Fig III.8).

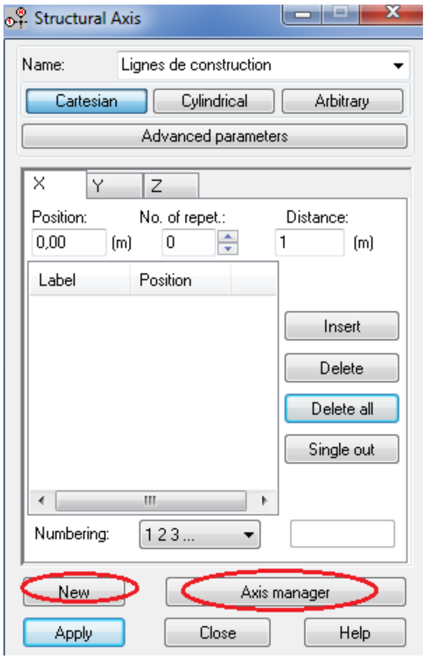


Figure III.8: Management of the construction lines.

III.3.4 Definition of sections for bar elements (columns and beams)

Click on the dropdown menu Structure -- characteristic -- bar profiles.

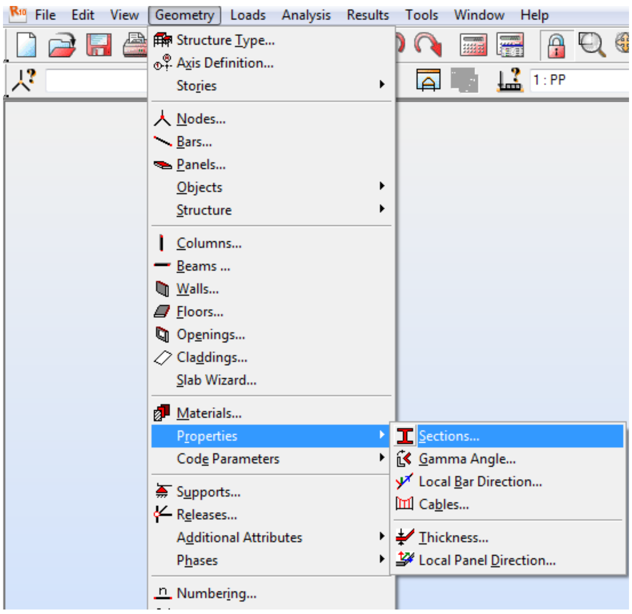


Figure III.9: Definition of the sections.

In the dialog box (profiles), click on (delete all unused sections) and then click on (new).

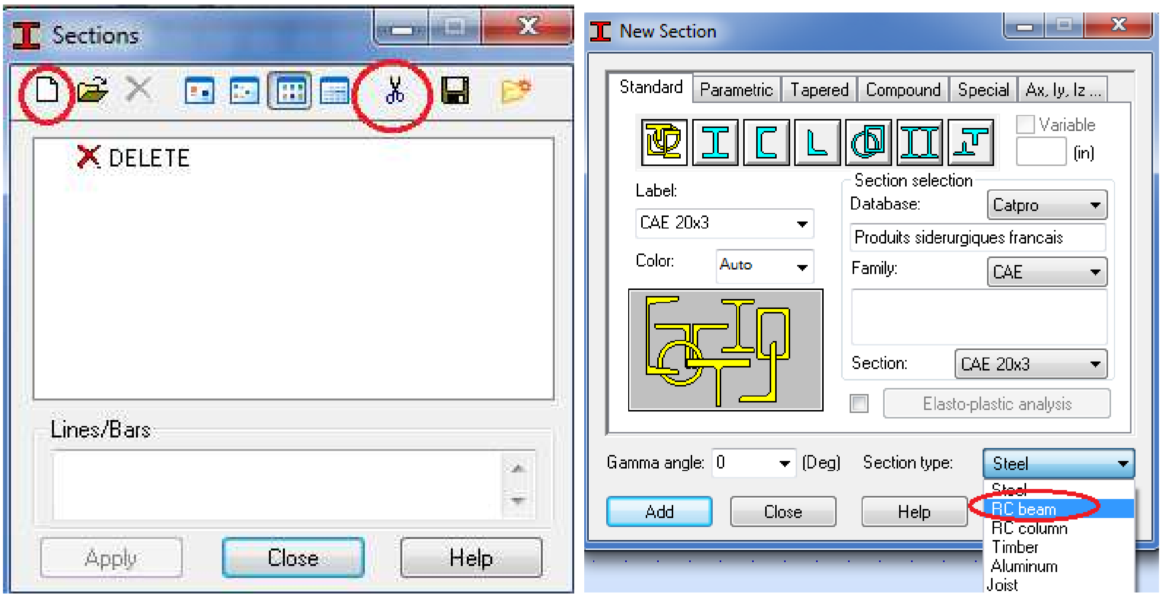


Figure III.10: Removing the default sections provided by the software.

In the dialog box (new section), click on the field (profile type) and select (beam BA).

Enter the name, color, and dimensions of the beam, then click "Add", (Fig III.11):

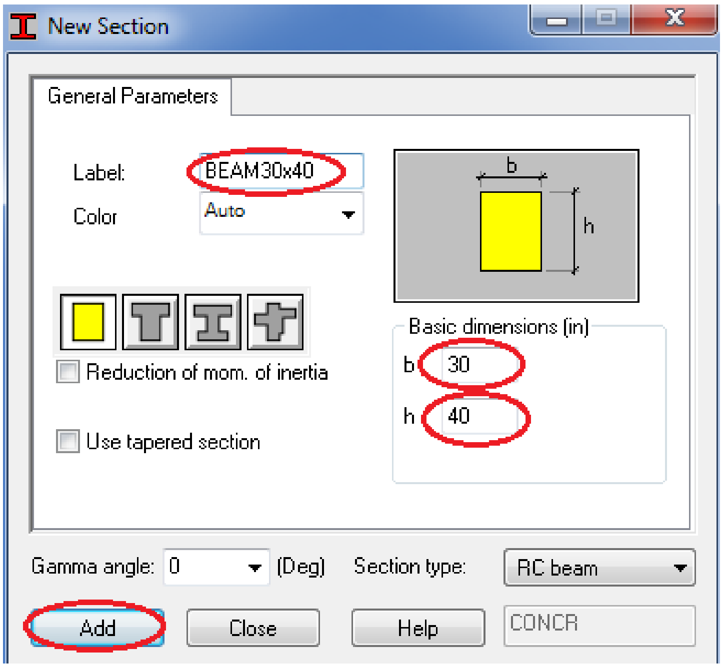


Figure III.11: Creating the beam sections.

Repeat the same operation to define the other sections of the beams and columns (PS 30x35) and (columns 30x40), (Fig III.12).

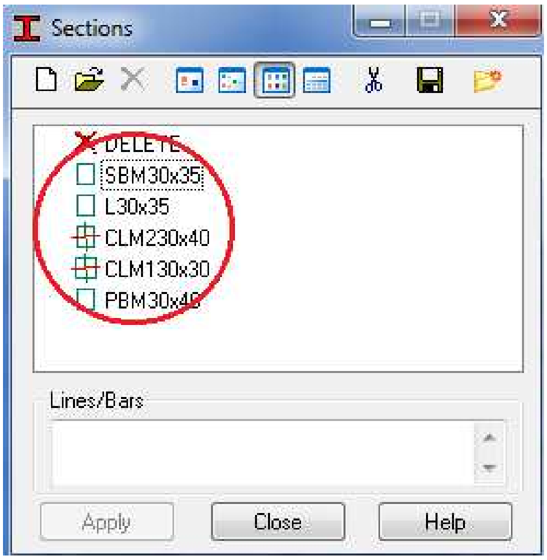


Figure III.12: Achievement of the section creation.

III.3.5 Structure definition

Activate the "View Manager" dialog box and go to the XY plane at a level of 3.20:

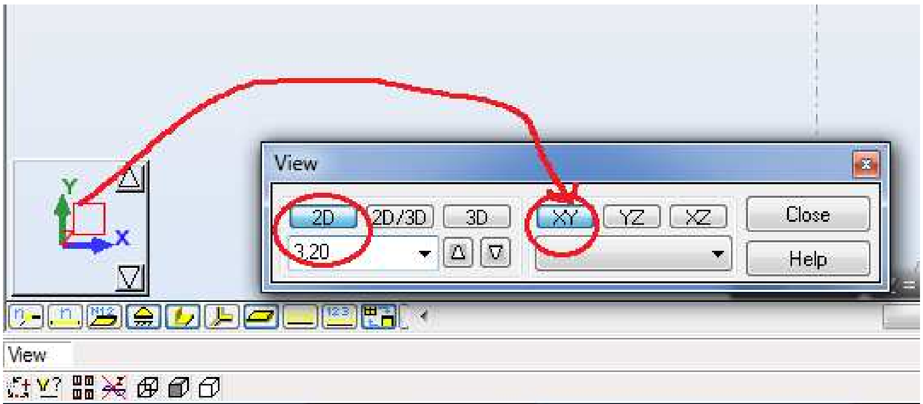


Figure III.13: Activation of the view manager.

To avoid modeling errors, disable the grid snap by clicking on the snap mode icon (located at the bottom left of the window), (Fig III.14):

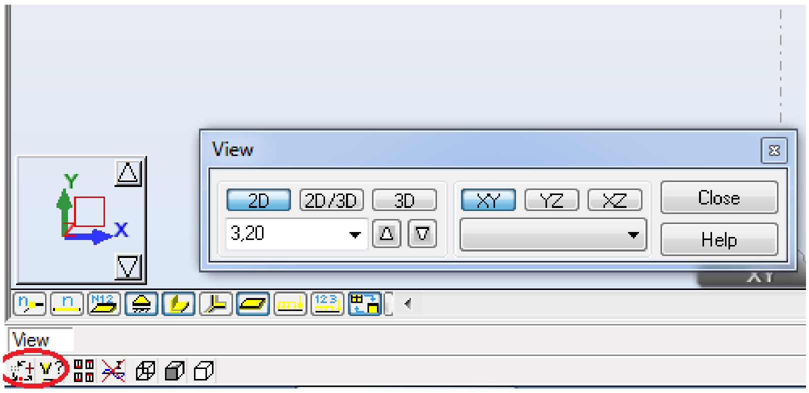


Figure III.14: Snap mode icon.

In the snap mode dialog box, disable the grid snap and click "Apply" and "Close", (Fig III.15):

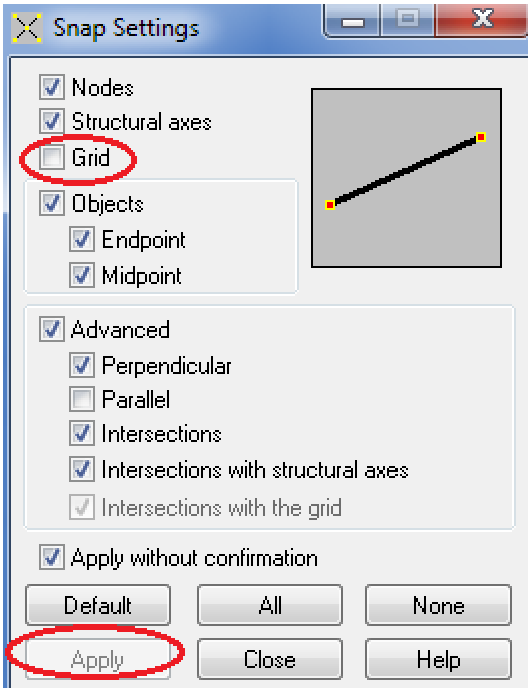


Figure III.15: Snap setting dialog box.

Now, click on the dropdown menu "Structure - Bars". The following dialog box opens:

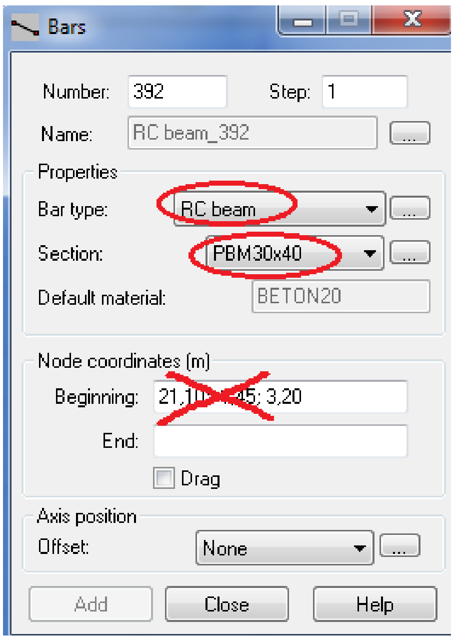


Figure III.16: Creation of the bar elements.

In the field (type), select beam BA, in the field (section), select (PP 30x40). Click on the field (origin) and start drawing the main beams. Using the same principle, we can draw all the main and secondary beams on floor level 3.20.

Now we will model the columns using the command (translation) with the option (stretched). First, we need to select the nodes on floor 3.20. Go to the drop-down menu Edit -- Special Selection -- Graphic Selection Filter, (Fig III.17):

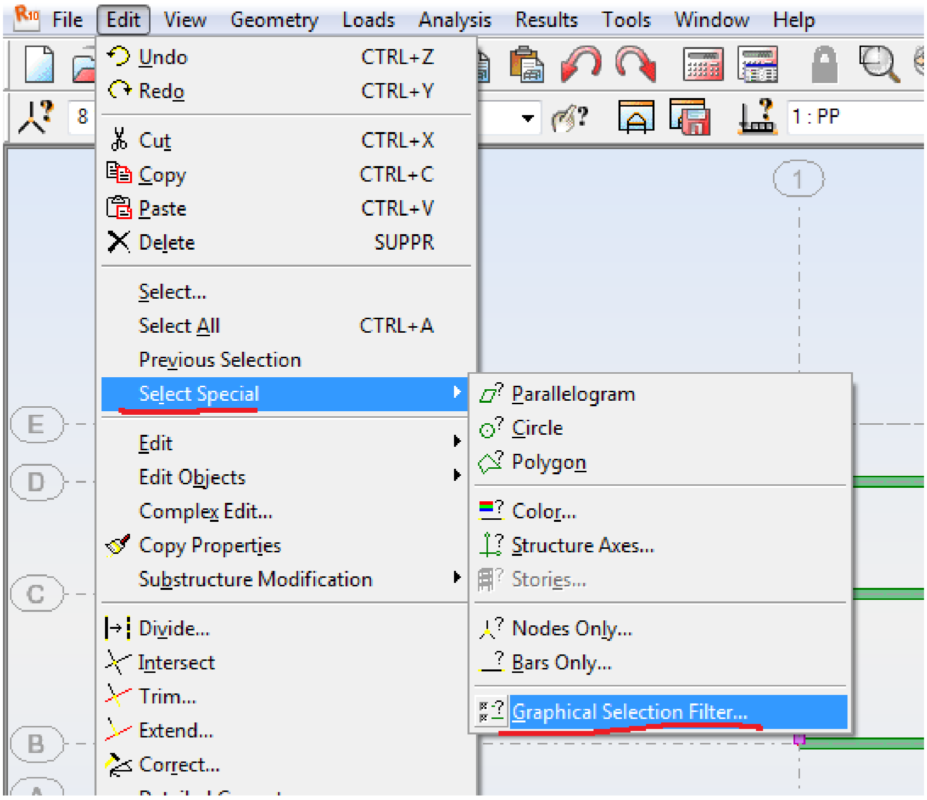


Figure III.17: Graphical selection toolbar.

In the dialog box (Graphic Selection Filter), uncheck all the boxes except for the box (node), (Fig III.18).

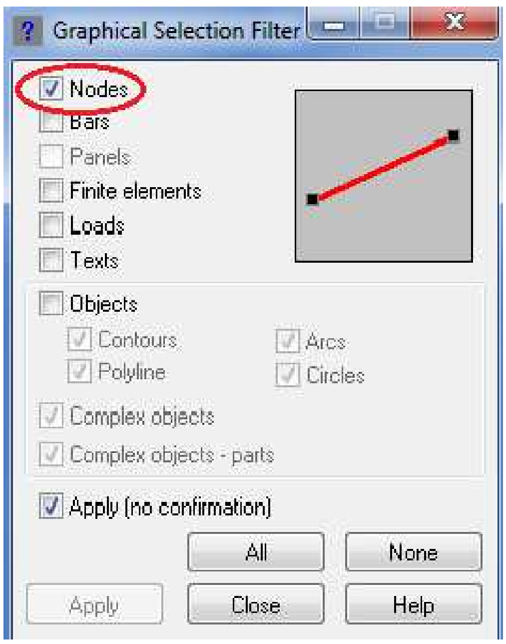


Figure III.18: Graphical selection filter dialog box.

Click on apply and close.

In the dialog box (Profiles), select (column 30x40) and close. Now select the entire structure, and you will notice that only the nodes are selected (the selection of other elements is disabled).

Go to the drop-down menu Edit transformation -- translation, (Fig III.19) :

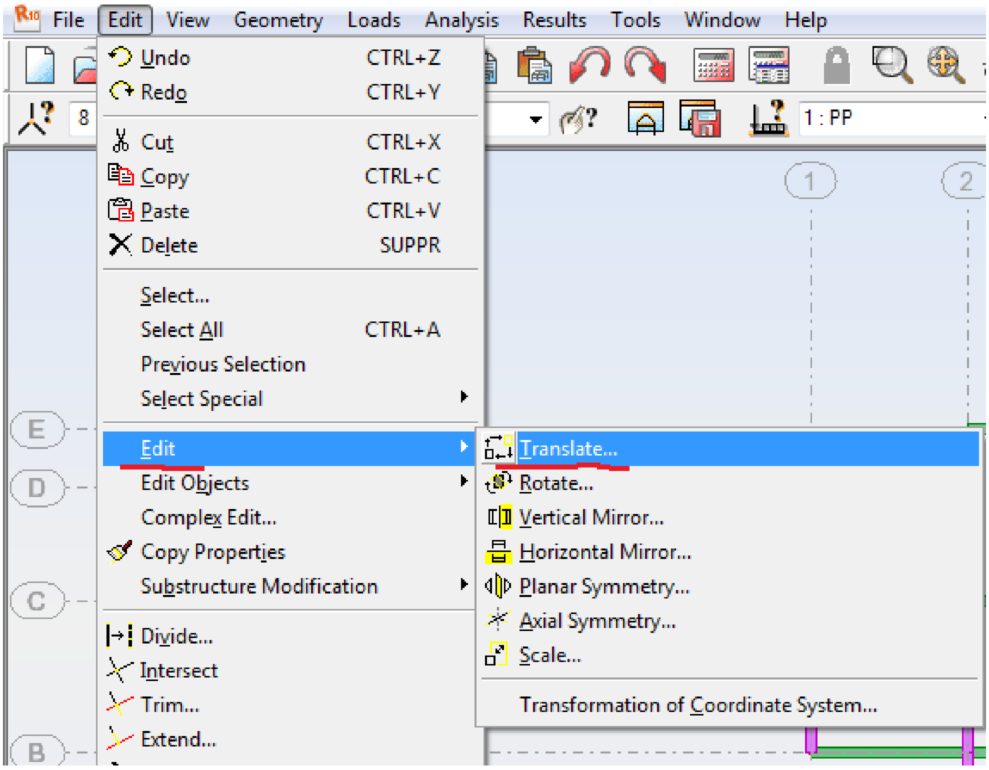


Figure III.19: Selection and displaying the edit toolbar.

Activate the 3D view and enter the value (0; 0; 3.20) in the dialog box (translation). Enable the option (stretched), (Fig III.20):

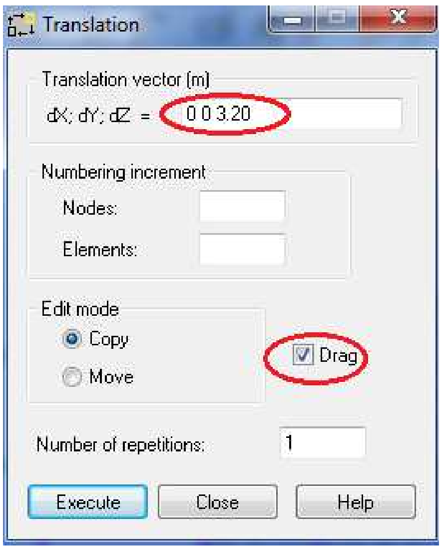


Figure III.20: The translation dialog box.

Click on (apply), and you will get the following result, (Fig III.21):

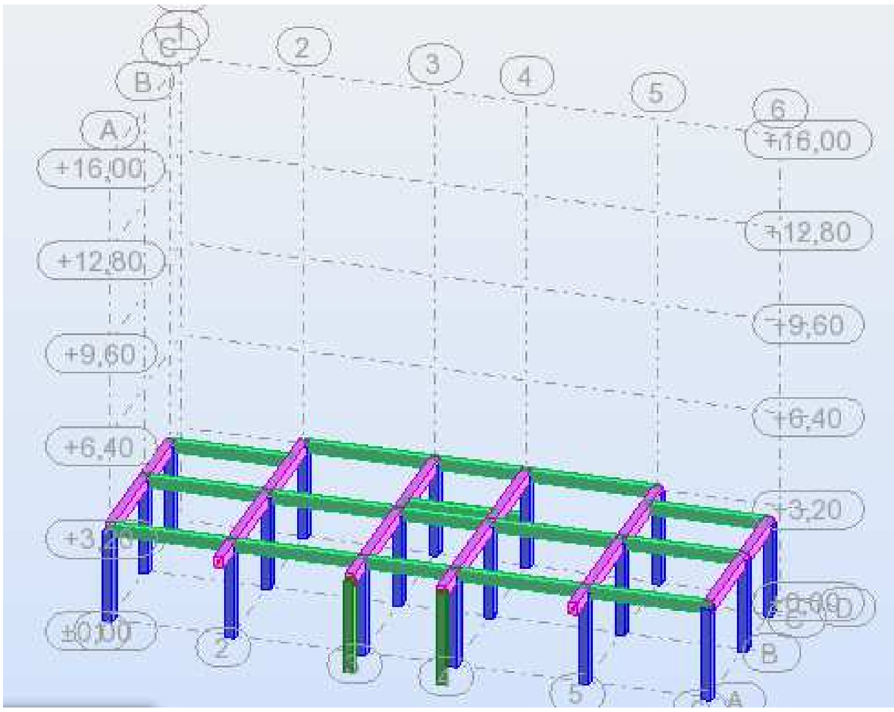


Figure III.21: 3D view of the constructed elements.

Go to the dialog box (Graphic Selection Filter) and enable all selections. Press (Ctrl+A) to select the entire structure. Go to the dialog box (translation) and make the following adjustments, (Fig III.22):

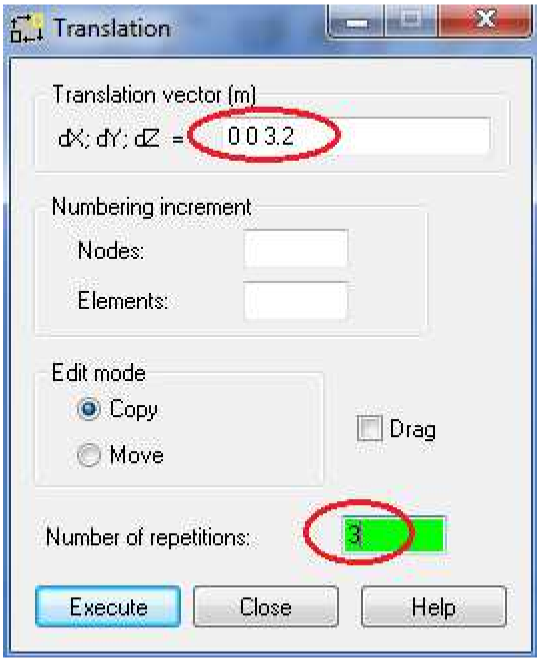


Figure III.22: Adjustments of the translation dialog box.

And you will get the following result:

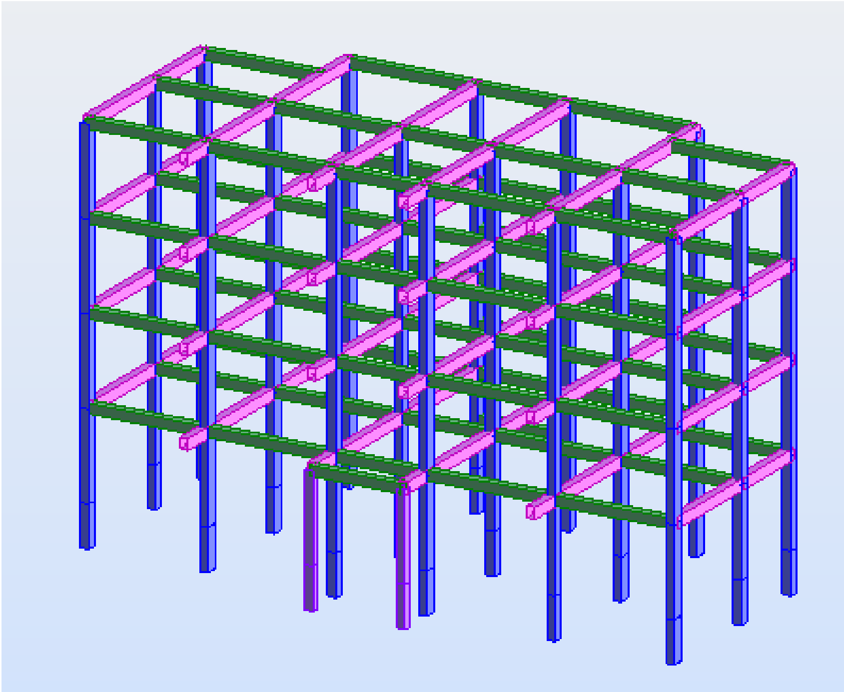


Figure III.23: 3D view of the obtained structure.

III.4 Loading

Click on the drop-down menu (Loading -- Load case), and you will have the dialog box (Load case). In this dialog box, we will define two types of load cases (Permanent load G and Live load Q), (Fig III.23):

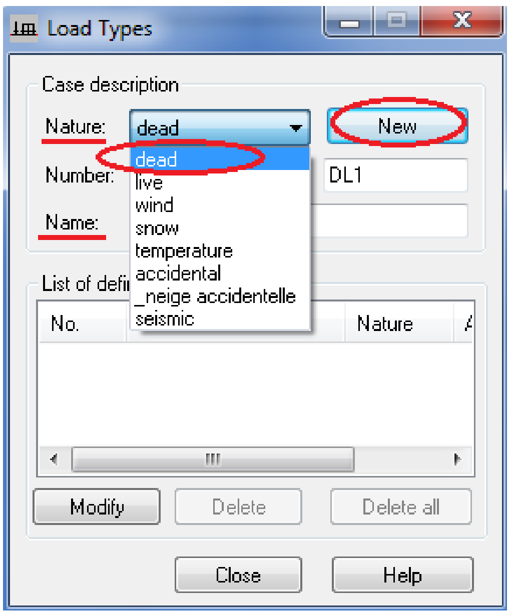


Figure III.24: Load dialog box.

The self-weight will be considered with the permanent load G. For seismic loads, they will be generated automatically by the software. Other loads such as wind and snow will be neglected.

III.4.1 Definition of Cladding

Click on the drop-down menu Structure -- Cladding:

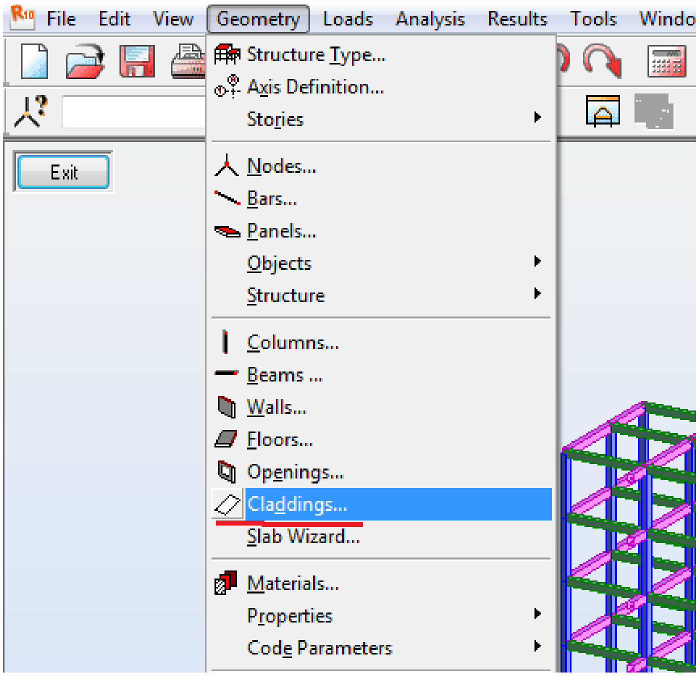


Figure III.25: Selection of the claddings toolbar .

In the dialog box (Cladding), define the number and direction of the cladding, and finally click on apply, (Fig III.26):

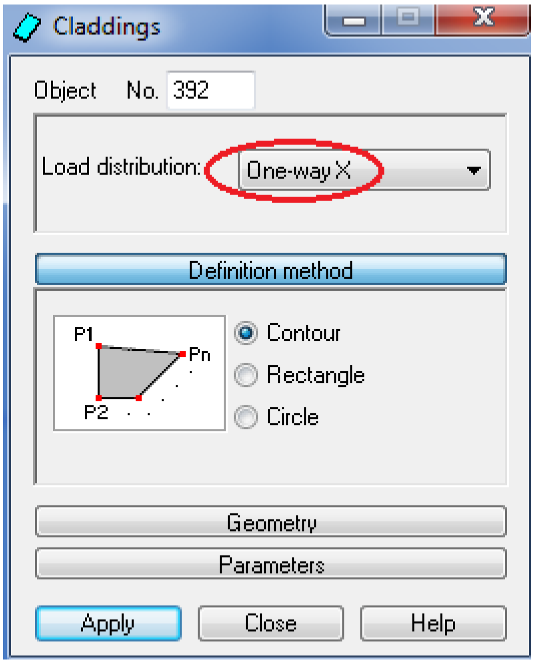


Figure III.26: Cladding dialog box.

III.4.2 Assignment of Loads

In the XY plane at level 3.20, go to the drop-down menu Loading -- Other loads -- Surface load on bar by 3D object, (Fig III.27):

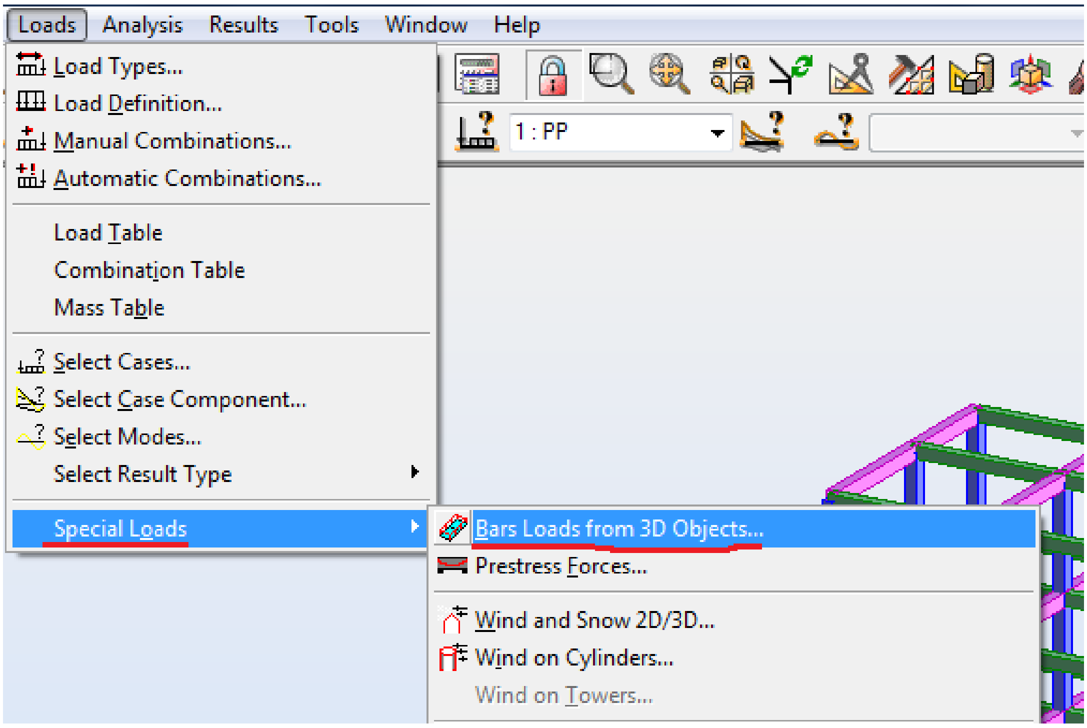


Figure III.27: Selection of the loading type.

In the dialog box (Load by object), click on (define) and draw the contour representing the floor, (Fig III.28):

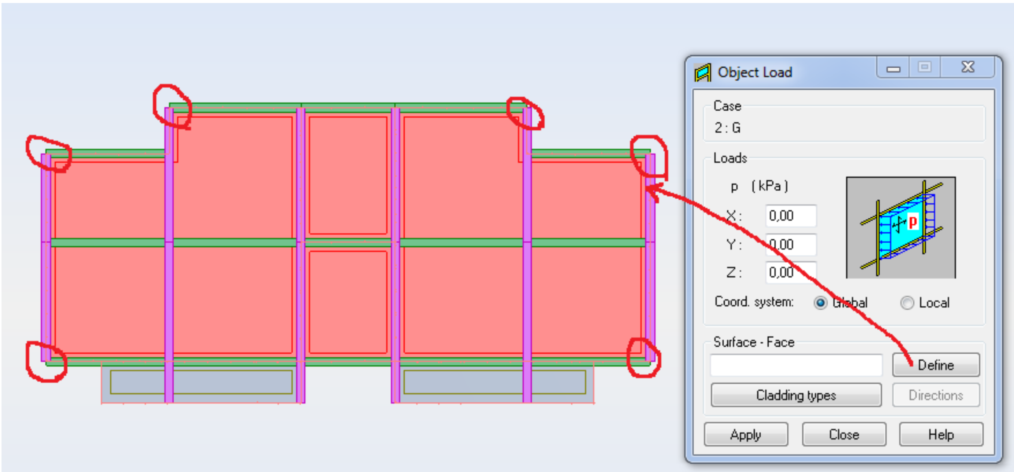


Figure III.28: Creation of the floor contour.

Note:

To avoid errors in the direction of the cladding, the first vector of the contour (line 1-2) must be parallel to the global X-axis.

In the (load case) zone, choose G and enter the value (-5.0 kPa) in the Z field of the dialog box (load by object), then click on (apply), (Fig III.29).

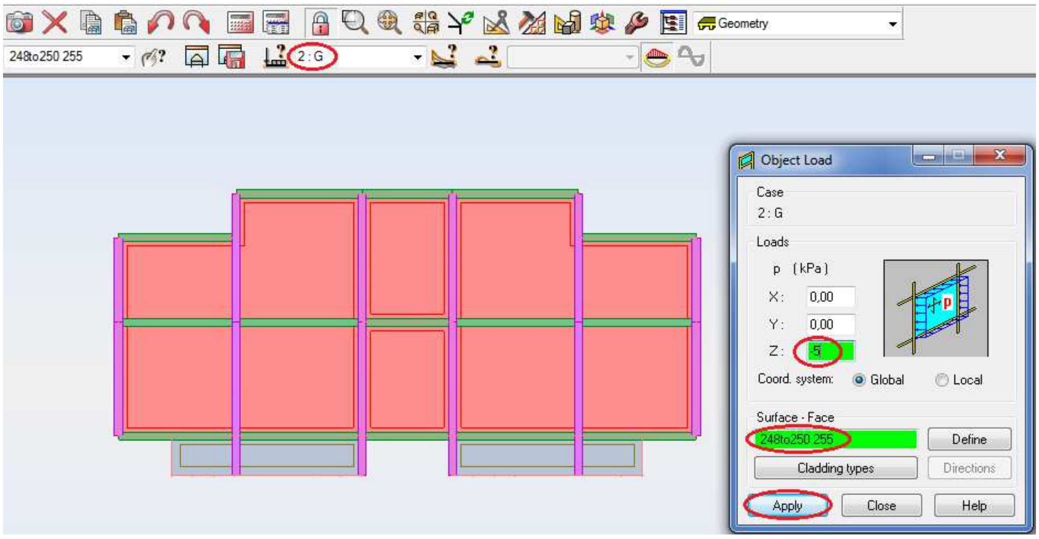


Figure III.29: Assignment of the loading.

Repeat the same operation with load case Q, entering the value (-1.5 kPa).

We need to repeat the same operation for all other levels except for level 16.00 (inaccessible terrace), where we need to replace the value (-5.0) with (-6.30) for the load G and the value (-1.5) with (-1.0) for the live load Q.

Thus, we obtain:

For the permanent load G:

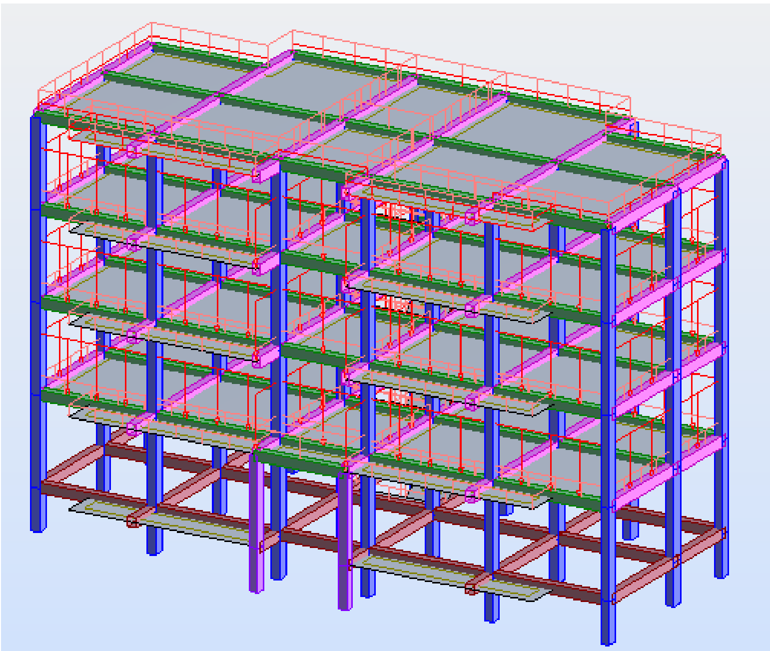


Figure III.30: Visualizing applied loads in the 3D view of the model.

III.4.3 Load on solid slabs

For solid slabs and stairs, we need to use the (define load) dialog box. Click on the dropdown menu Load -- Define load, (Fig III.31):

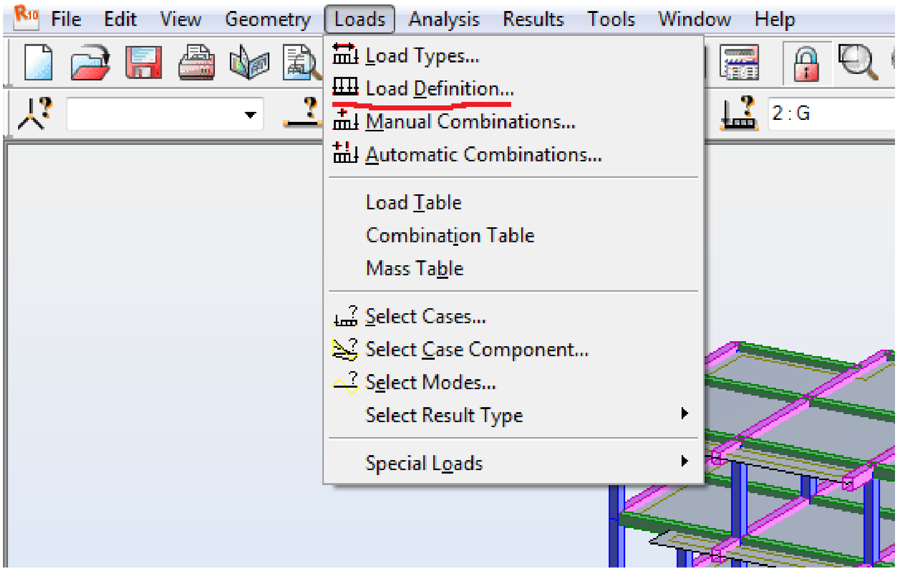


Figure III.31: Visualization of the loads menu.

In the (load) dialog box, click on (surface) and then click on (uniform surface load), (Fig III.32):

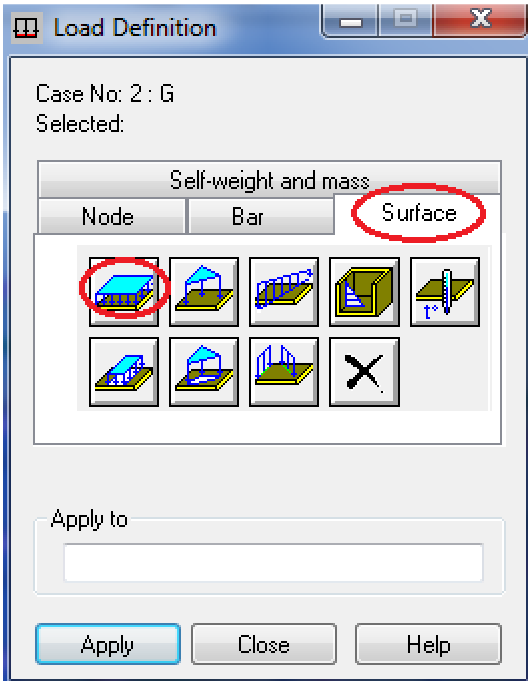


Figure III.32: Load definition dialog box.

In the (uniform surface load) dialog box, enter the value (-3.5 kPa) which represents the live load on the balconies. Click on (add), (Fig III.33):

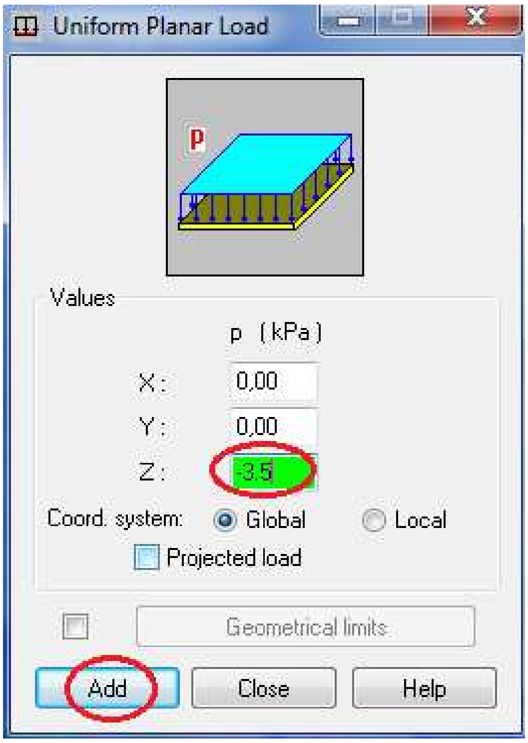


Figure III.33: Application of uniform planar load.

Now, in the (load case) area, select load case Q, and in the (apply to) field of the (load) dialog box, enter the names of all panels representing the balconies and click on apply, (Fig III.34):

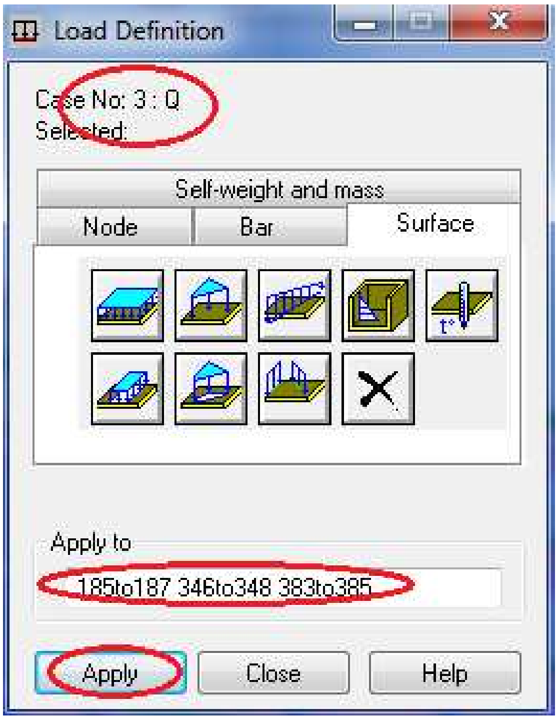


Figure III.34: Application of Q load on the balconies.

We need to do the same to define the loads on all solid slabs and stairs.

III.5 Mesh generation

Select all panels, then go to the dropdown menu (Analysis -- Calculation Model -- Mesh Options), (Fig III.35):

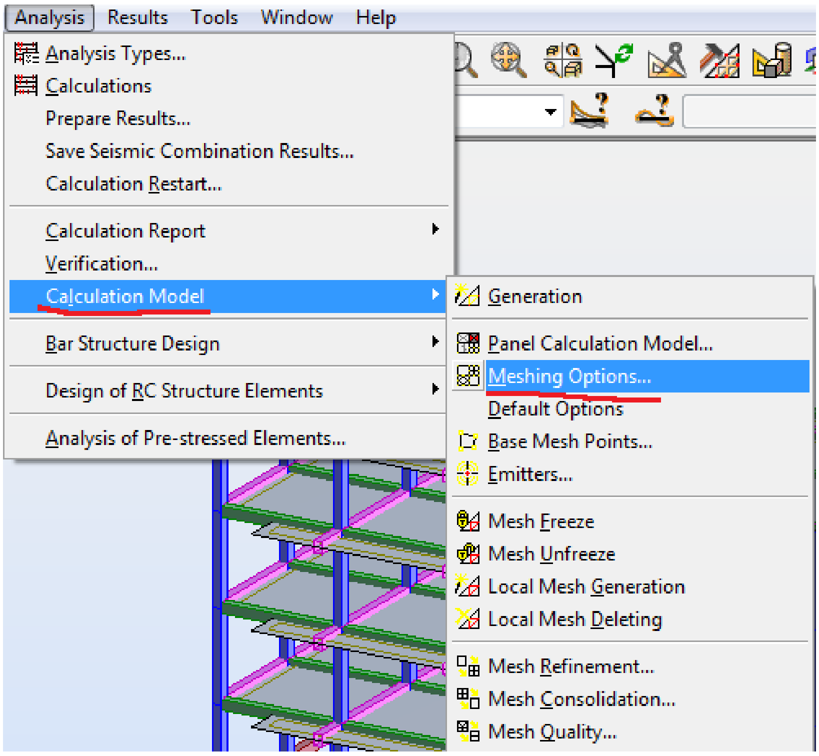


Figure III.35: Meshing options toolbar.

In the (mesh options) dialog box, make the following settings, (Fig III.36):

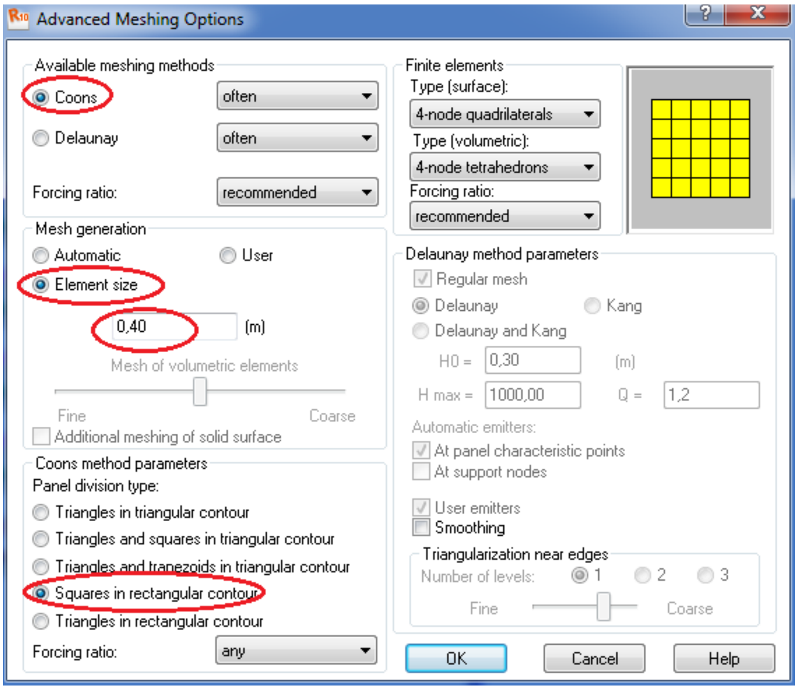


Figure III.36: Setting of the mashing options dialog box.

Click on OK, then go to the dropdown menu (Analysis -- Calculation Model -- Generate), (Fig III.37):

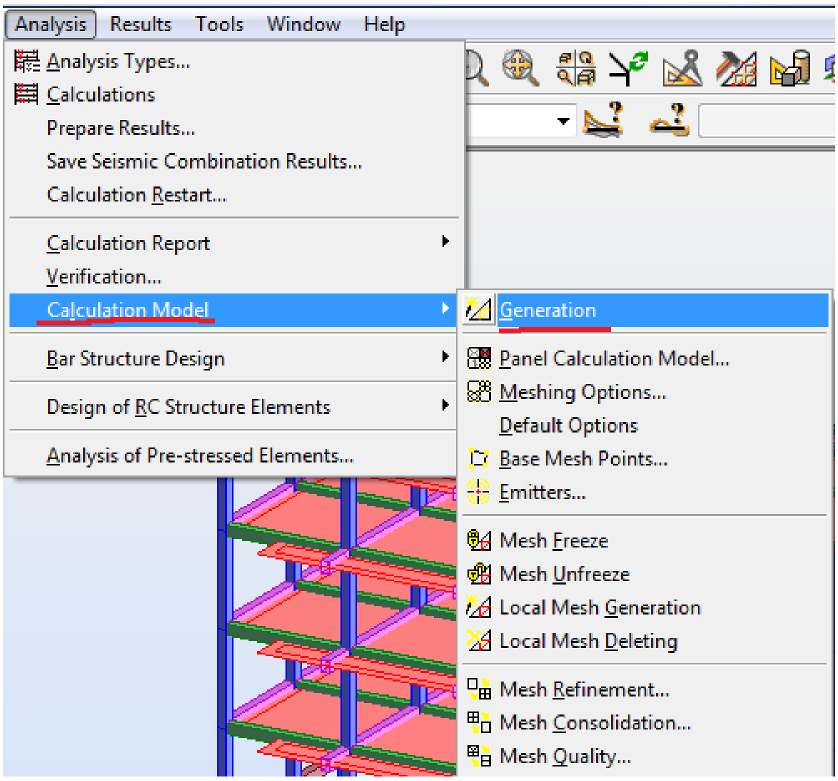


Figure III.37: Mesh generation toolbar.

Mesh generation takes some time, and you will have the following result at the end, (Fig III.38):

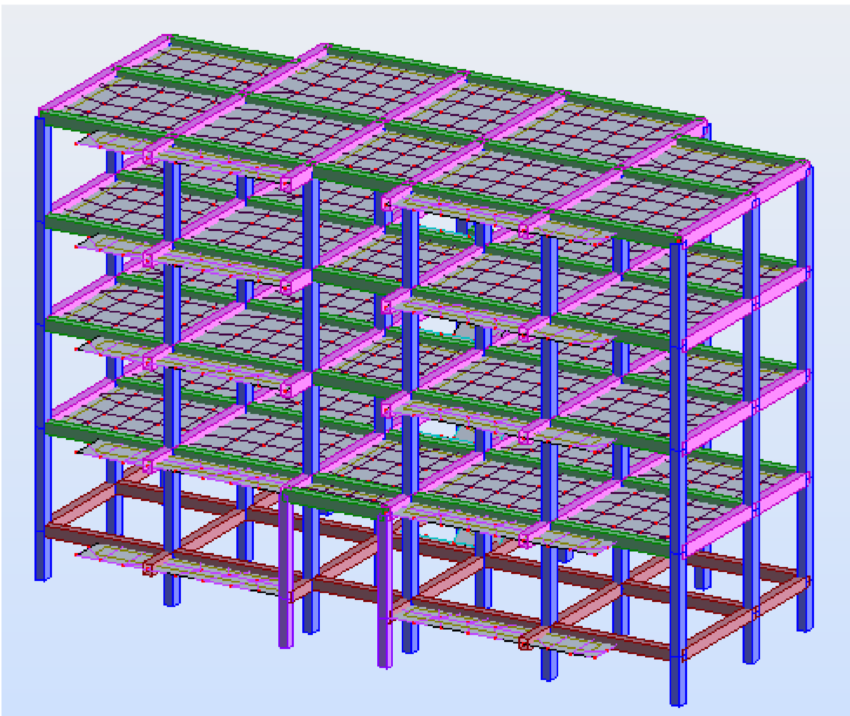


Figure III.38: 3D view of the obtained structure.

III.6 Definition of supports

To avoid errors related to support definition, you need to disable the selection of all objects and leave only the selection of nodes enabled, (Fig III.39):

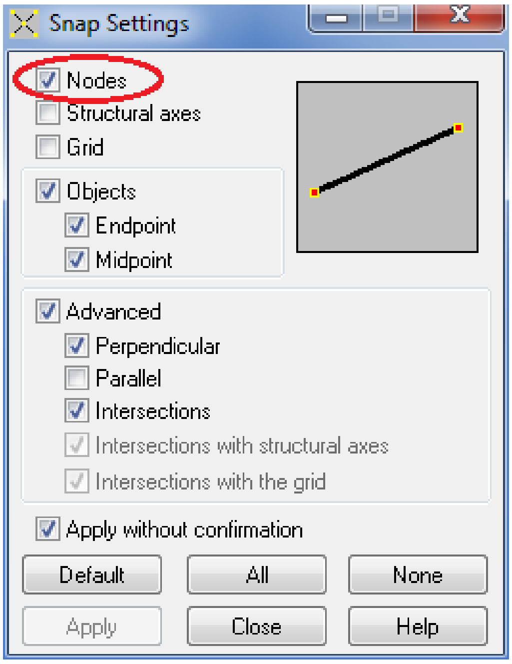


Figure III.39: Snap setting dialog box.

Click on the dropdown menu (Structure -- Supports), (Fig III.40):

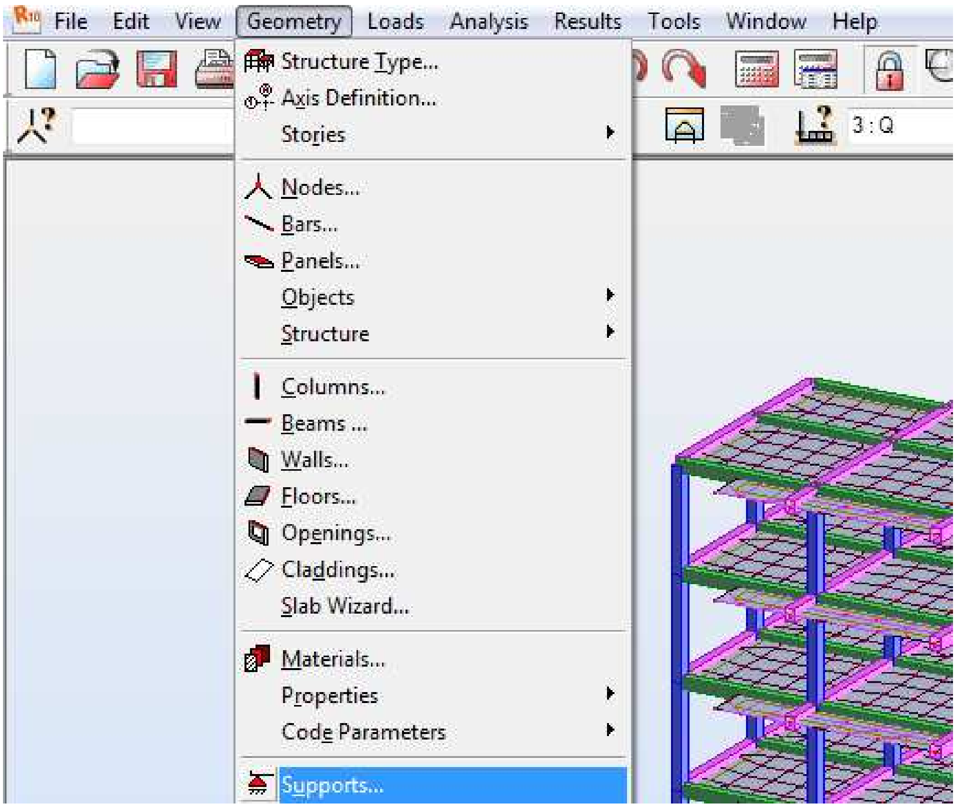


Figure III.40: Supports icon in the geometry menu.

In the (Supports) dialog box, make the following settings, (Fig III.41):

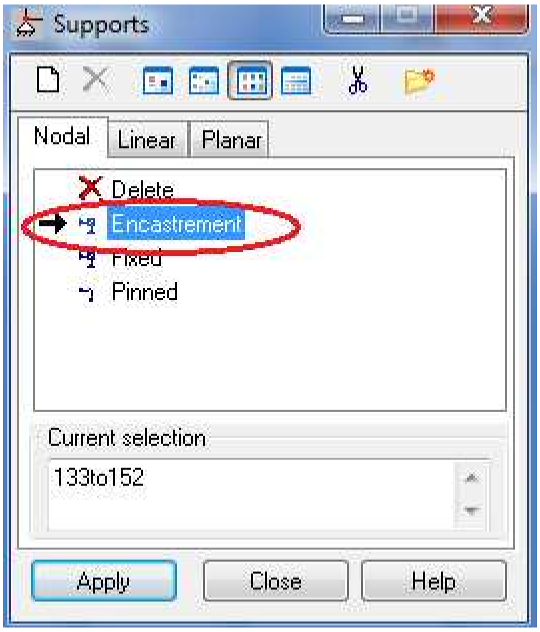


Figure III.41: Setting of the supports dialog box.

Make sure that for the support type (fixed support), all translations and rotations are locked.

In the (Current selection) area, select all nodes at level 0.00 and click on (Apply). You will notice that the fixed support symbol is displayed on all nodes at level 0.00, (Fig III.42).

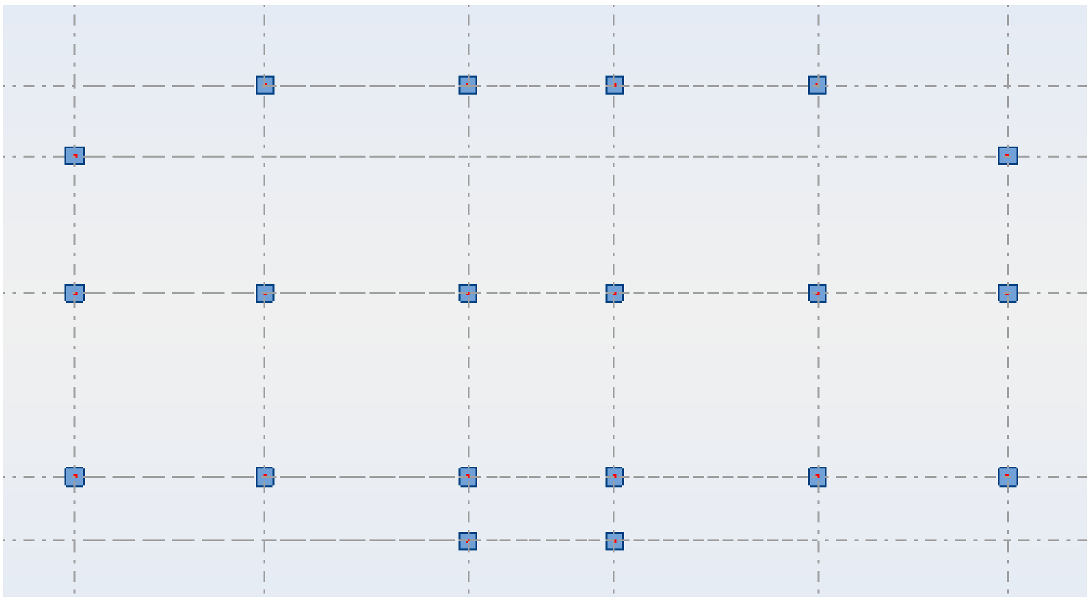


Figure III.42: Plan view of the support.

III.7 Modal and seismic analysis

To declare a modal analysis, click on the dropdown menu Analysis, then Analysis Types to display the calculation options dialog box:

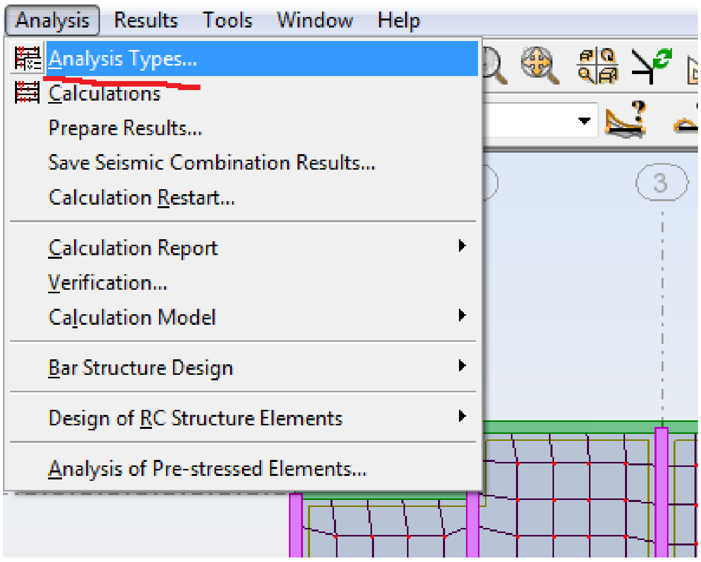


Figure III.43: Contents of the analysis menu.

In the (calculation options) dialog box, click on New, (Fig III.44):

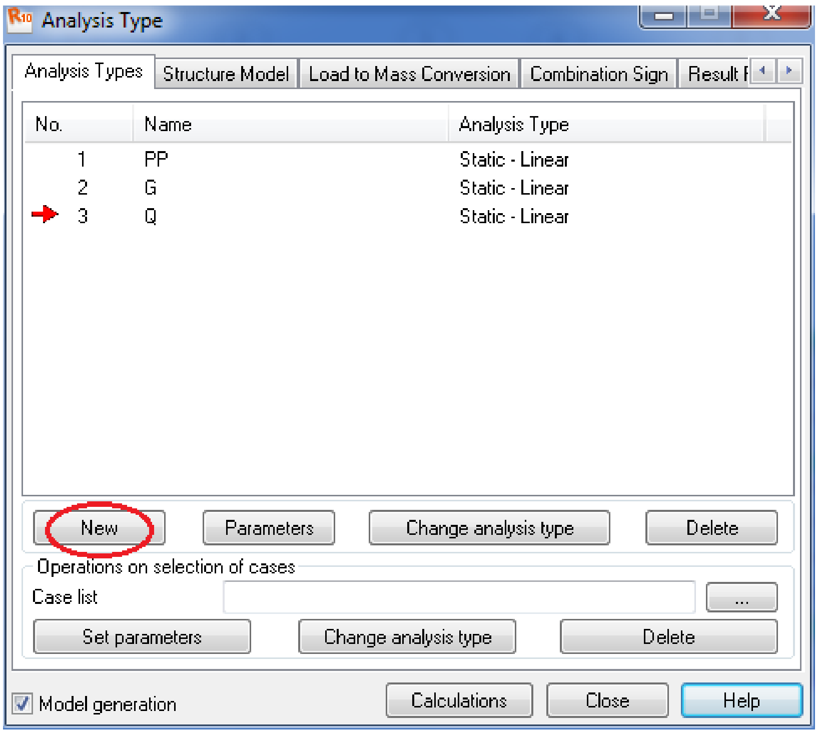


Figure III.44: Definition of a new load case.

Select (modal analysis type) and click OK. In the (Modal Analysis Parameters) dialog box, make the following settings, (Fig III.45):

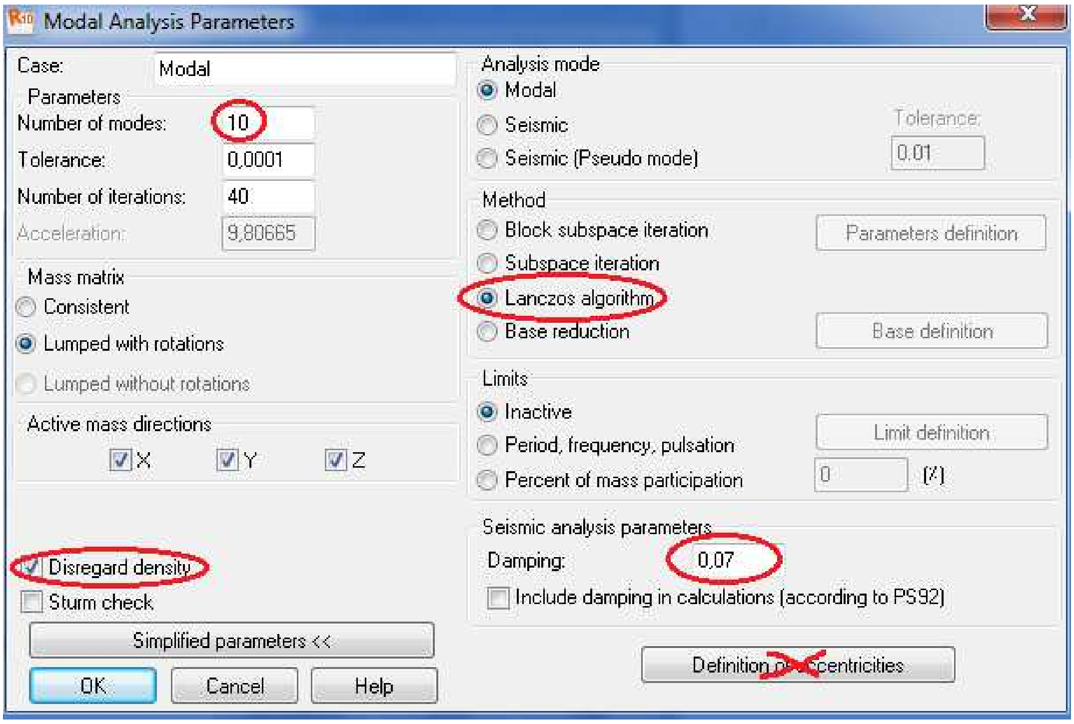


Figure III.45: Setting of the modal analysis.

Before exiting the (Modal Analysis Parameters) dialog box, click on (eccentricity) and enter the following values, (Fig III.46):

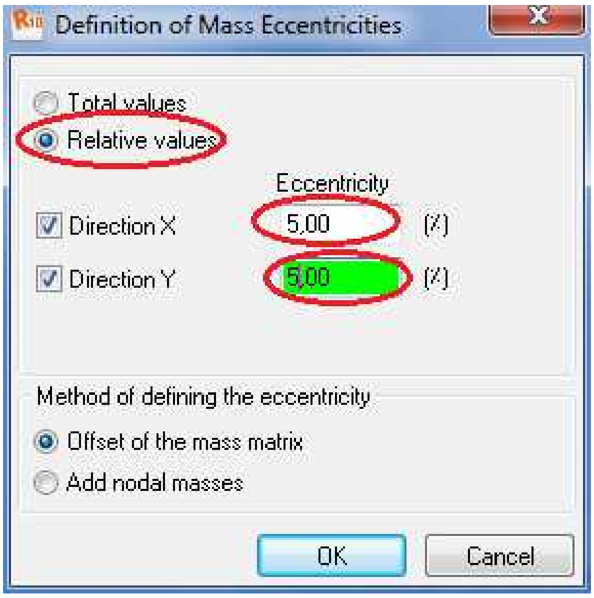


Figure III.46: Definition of mass eccentricities.

Click OK, and you will notice the display of a new load case called "modal".

Click again on (New), choose (seismic), and select (RPA 99 (2003) (Algeria)), (Fig III.47):

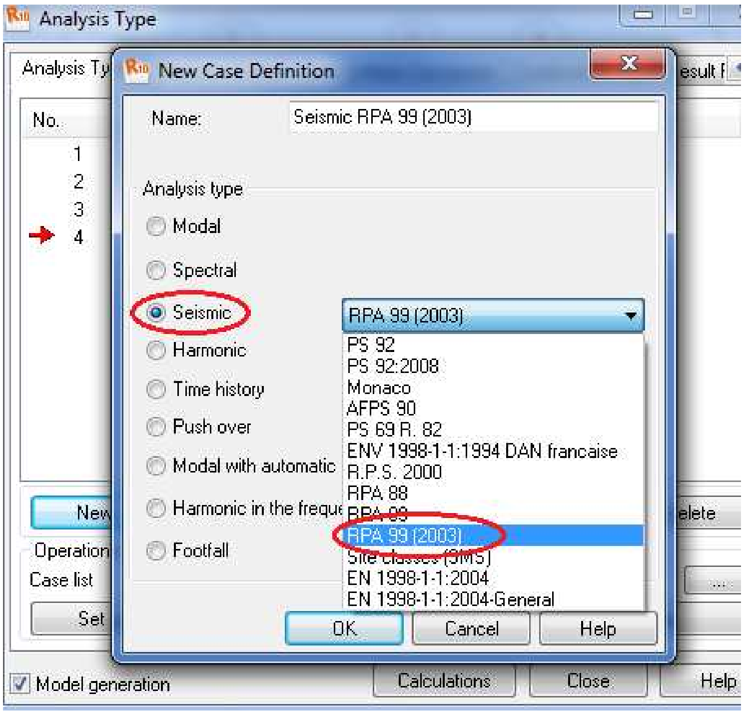


Figure III.47: Definition of the seismic load.

In the (RPA99 parameters) dialog box, select the following options, (Fig III.48):

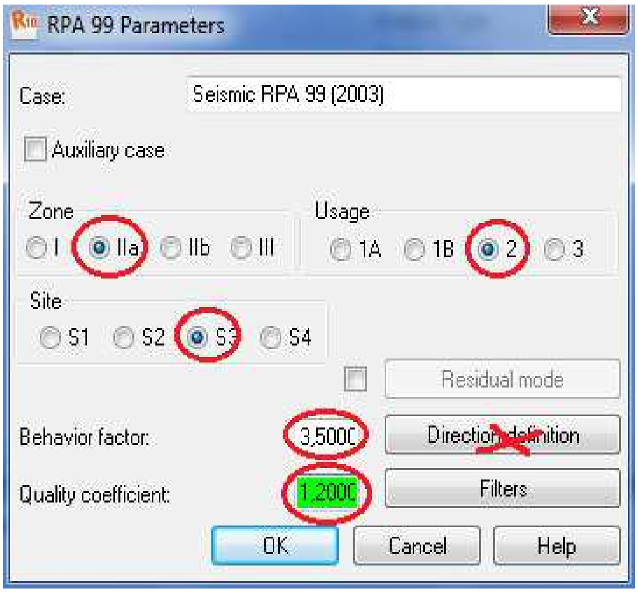


Figure III.48: Setting of the seismic parameters.

Before exiting the (RPA99 parameters) dialog box, click on (Direction Definition) and make the following settings, (Fig III.49):

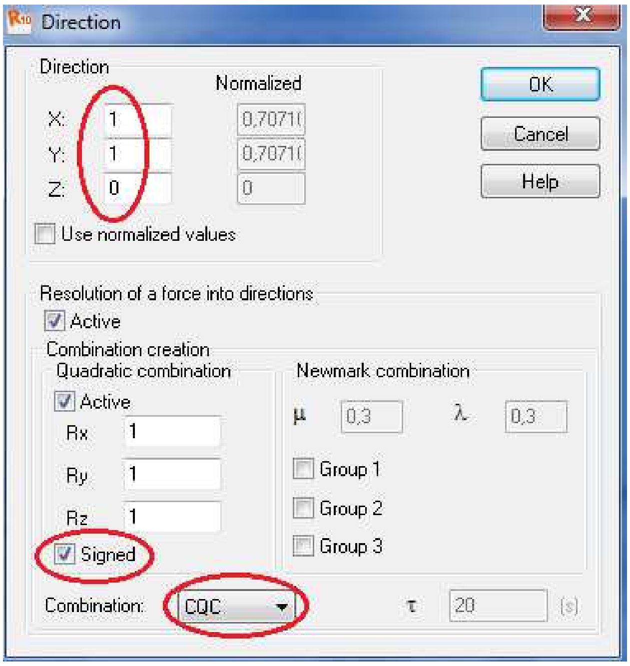


Figure III.49: Setting of the seismic directions.

Click OK, and you will notice the display of 2 seismic load cases, one along the X-axis and the other along the Y-axis, (Fig III.50):

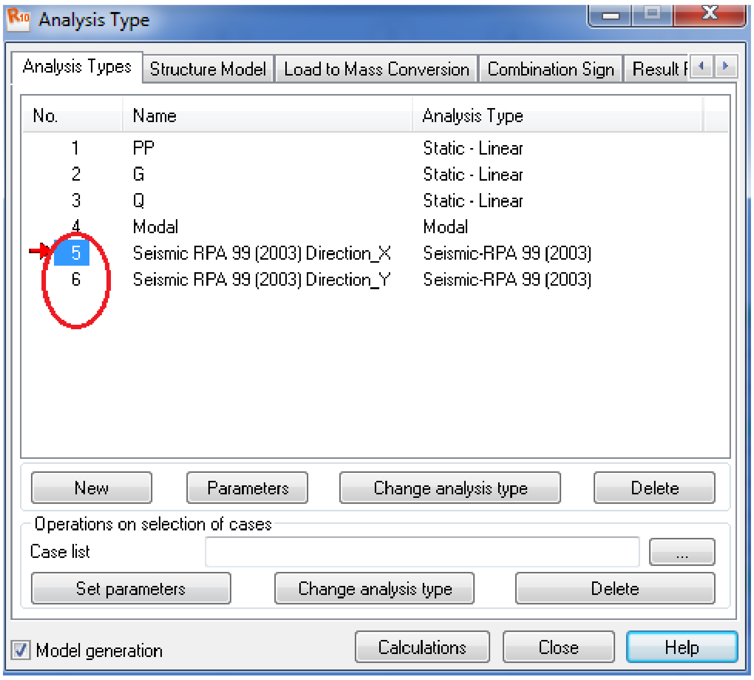


Figure III.50: Creation of the seismic loads.

III.8 Combination of load cases

For our example, we will define the combination at ultimate limit state 1.35 G + 1.5 Q and the combination at serviceability limit state G + Q.

To define the load case combinations, click on the dropdown menu Loads, then Manual Combinations, (Fig III.51):

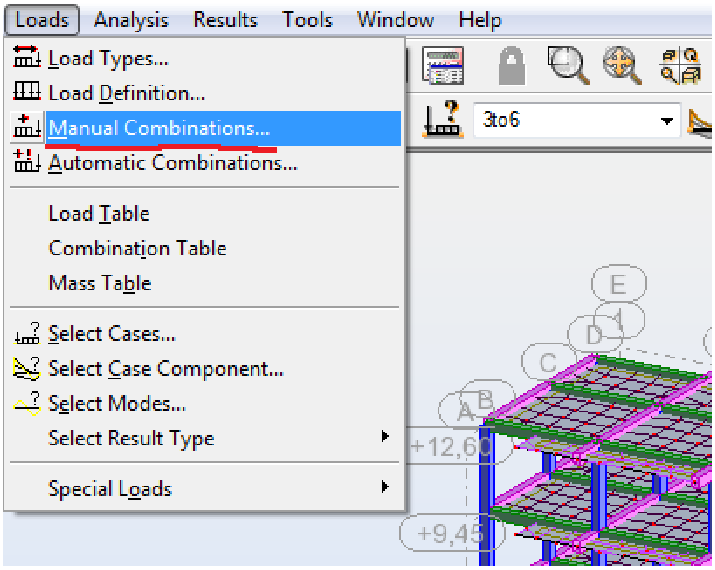


Figure III.51: Contents of the loads menu.

The following dialog box opens, (Fig III.52):

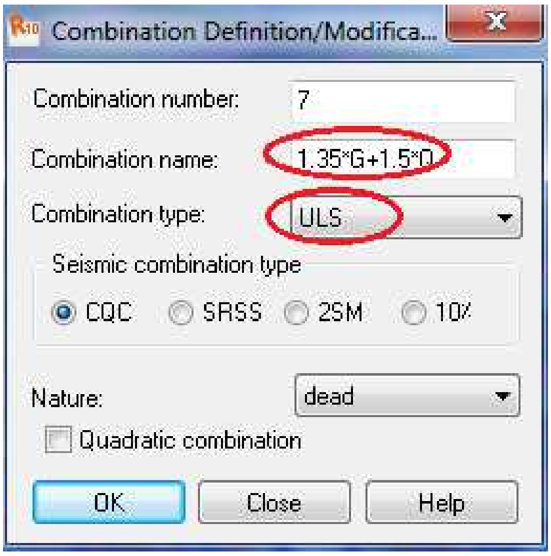


Figure III.52: The combination dialog box.

Choose the combination type and give the desired name to the combination you are going to define, for example, the name 1.35 G + 1.5 Q, and click OK.

In the opened dialog box, define the combination using the previously defined load cases, (Fig III.53):

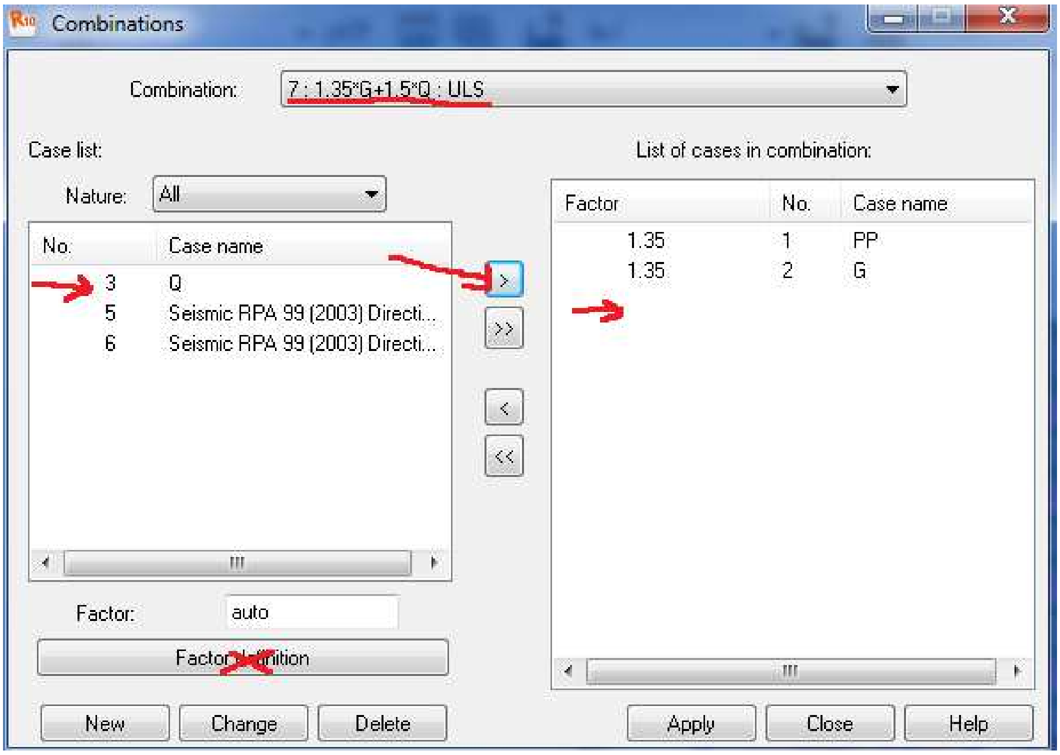


Figure III.53: Creation of the ULS combination.

If the coefficients you want to apply are different from the automatic coefficients of the defined combination, you can define them by clicking on Define Coefficients.

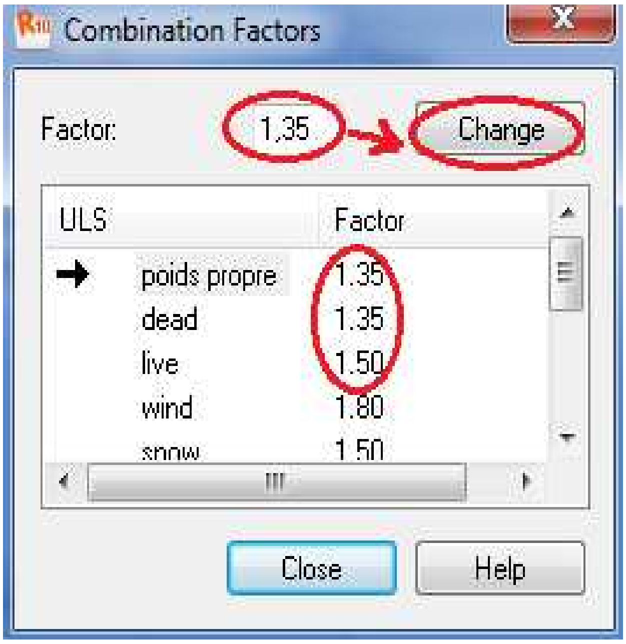


Figure III.54: Definition and modifying the combination factors.

Enter the desired coefficient value in the Coefficient field and click Modify, (Fig III.53).

At the end of this operation, click Apply to save the combination. You can define another combination by clicking New and repeating the same steps, changing the coefficients for each combination.

III.9 Analysis and analysis results

III.9.1 Calculation and analysis

Now that we have finished modeling our structure, we proceed to the calculation and analysis of this structure under the defined loading.

Before running the calculation, we need to check the structure for modeling errors and disconnected bars. To do this, click on the Analysis menu ► Verify Structure, (Fig III.55).

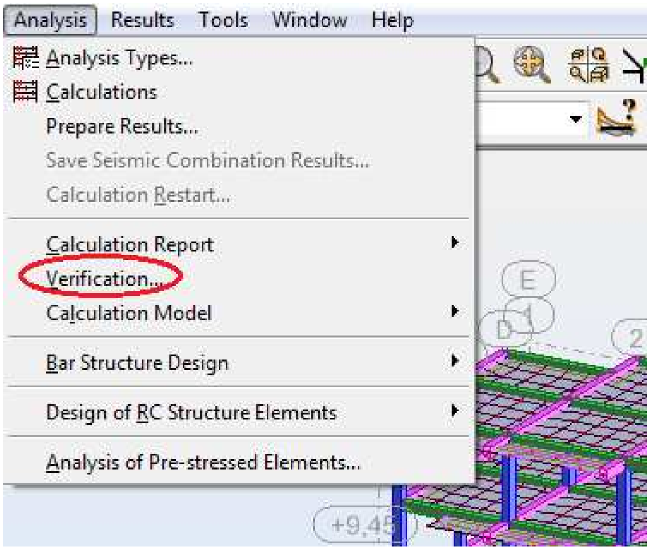


Figure III.55: Verification of the structure.

In the dialog box, the error message indicates the error and the object related to this error as shown in the figure III.56:

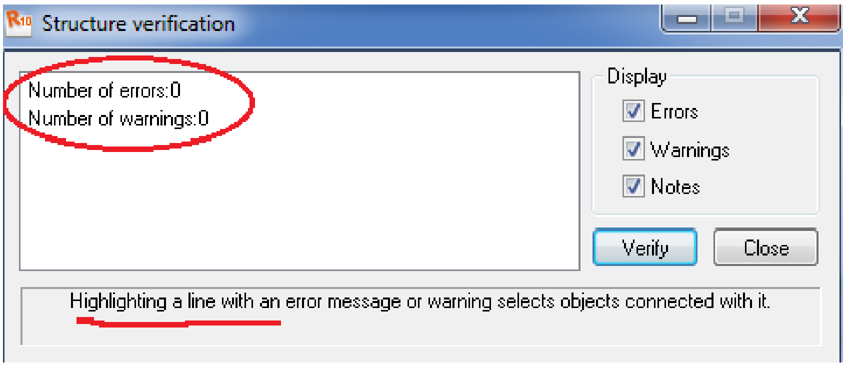


Figure III.56: Structure verification dialog box.

To start the calculation, click on the Analysis menu, then Calculate (Analysis ► Calculate), (Fig III.57).

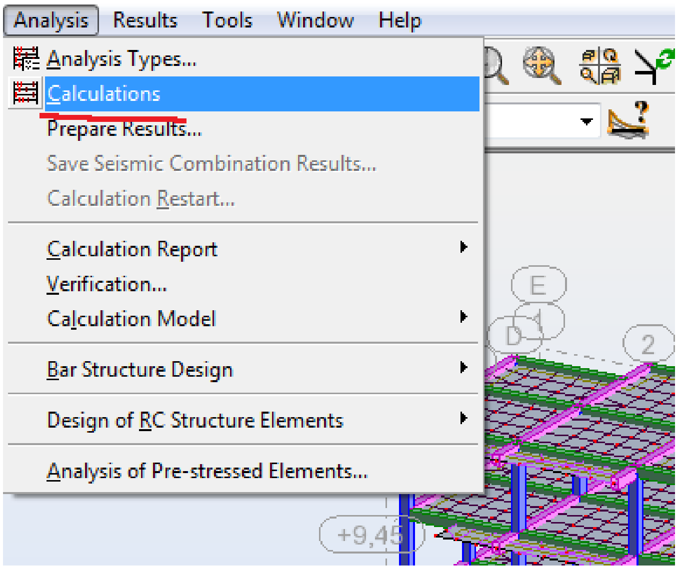


Figure III.57: Initiating of the calculation.

III.9.2 Analysis results

To display the desired results (diagrams, reactions, displacements, stresses, deformations, etc.), click on the "Results" menu. If you want to display them in tabular form, simply right-click and choose "Tables", (Fig III.58).

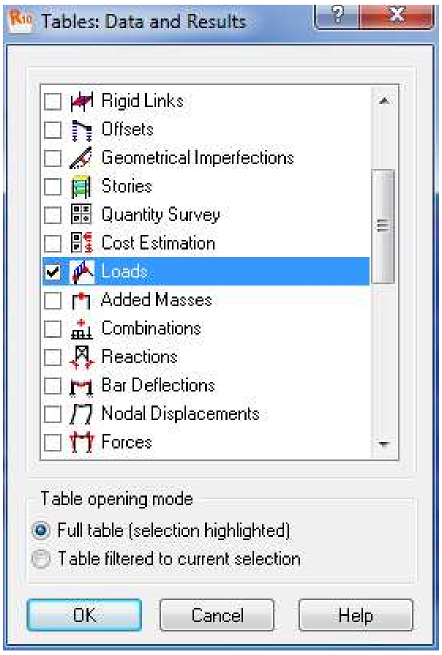


Figure III.58: Displaying tabular results.

a. Results Verification

Right-click and then click on "Tables", check the "Eigenmode" box, and the results related to modal analysis will be displayed.

Case/Mode	Frequency (Hz)	Period (sec)	Rel.mas.UX (%)	Rel.mas.UY (%)	Rel.mas.UZ (%)	Cur.mas.UX (%)	Cur.mas.UY (%)	Cur.mas.UZ (%)	Total mass UX (kg)	Total mass UY (kg)	Total mass UZ (kg)
6/ 1	1,28	0,78	66,00	0,01	0,00	66,00	0,01	0,00	980429,17	980429,17	980429,17
6/ 2	1,46	0,68	66,01	83,79	0,00	0,01	83,78	0,00	980429,17	980429,17	980429,17
6/ 3	1,55	0,64	85,13	83,89	0,00	19,12	0,10	0,00	980429,17	980429,17	980429,17
6/ 4	3,73	0,27	93,01	83,89	0,00	7,88	0,00	0,00	980429,17	980429,17	980429,17
6/ 5	4,27	0,23	93,01	93,02	0,04	0,00	9,13	0,04	980429,17	980429,17	980429,17
6/ 6	4,62	0,22	93,42	93,03	0,04	0,41	0,00	0,00	980429,17	980429,17	980429,17
6/ 7	6,23	0,16	95,66	93,03	0,04	2,24	0,00	0,00	980429,17	980429,17	980429,17
6/ 8	7,40	0,14	95,67	95,41	0,09	0,01	2,38	0,05	980429,17	980429,17	980429,17
6/ 9	8,02	0,12	95,71	95,41	0,09	0,04	0,00	0,00	980429,17	980429,17	980429,17
6/ 10	8,77	0,11	96,14	95,41	0,09	0,43	0,00	0,00	980429,17	980429,17	980429,17

Tableau III.1: Results of the modal analysis.

b. Reaction Verification

Perform the same previous operation by checking "Reaction".

	FX (kN)	FY (kN)	FZ (kN)	MX (kNm)	MY (kNm)	MZ (kNm)
MAX	6,48	32,67	1296,91	9,27	5,32	0,56
Node	136	145	145	138	136	145
Case	4 (C)	4 (C)	4 (C)	4 (C)	4 (C)	4 (C)
MIN	-10,98	-12,89	13,90	-17,04	-6,45	-0,77
Node	145	138	142	145	145	147
Case	4 (C)	4 (C)	3	4 (C)	4 (C)	4 (C)

Tableau III.2: Reactions results.

c. Node Displacement Verification

Perform the same previous operation by checking "Node Displacements".

	UX (cm)	UY (cm)	UZ (cm)	RX (Rad)	RY (Rad)	RZ (Rad)
MAX	0,0	0,1	0,0	0,001	0,002	0,000
Node	244	338	325	464	302	418
Case	4 (C)	4 (C)	3	4 (C)	4 (C)	4 (C)
MIN	-0,0	-0,1	-0,5	-0,001	-0,002	-0,000
Node	1270	108	361	330	277	252
Case	4 (C)	4 (C)	4 (C)	4 (C)	4 (C)	4 (C)

Tableau III.3: Results in term of displacements.

d. Deflection Verification

Perform the same previous operation by checking "Bar Deflection".

	UX (cm)	UY (cm)	UZ (cm)
MAX	0,0	0,1	0,0
Node	244	338	325
Case	4 (C)	4 (C)	3
MIN	-0,0	-0,1	-0,5
Node	1270	108	361
Case	4 (C)	4 (C)	4 (C)

Tableau III.4: Results in term of deflections.

e. Verification of Bar Forces

If you want to display internal forces in the columns, select them and choose the combination for which you want to obtain the results.

	FX (kN)	FY (kN)	FZ (kN)	MX (kNm)	MY (kNm)	MZ (kNm)
MAX	1296,91	12,84	150,50	11,77	53,85	20,84
Bar	307	221	359	371	213	196
Node	145	92	22	16	114	103
Case	4 (C)	4 (C)	4 (C)	4 (C)	4 (C)	4 (C)
MIN	-38,57	-14,19	-130,19	-12,00	-99,84	-20,86
Bar	357	13	358	371	359	221
Node	16	17	22	22	22	118
Case	4 (C)	4 (C)	4 (C)	4 (C)	4 (C)	4 (C)

Tableau III.5: Results in term of forces.

f. Displaying Bar Force Diagrams

In the "Results" menu, select "Diagram" and click on the "Parameter" box to adjust the display of the diagrams. Then make your choices in the different tabs (NTM, Deformation, Stress, Reactions, ...), (Fig III.59) and (Fig III.60).

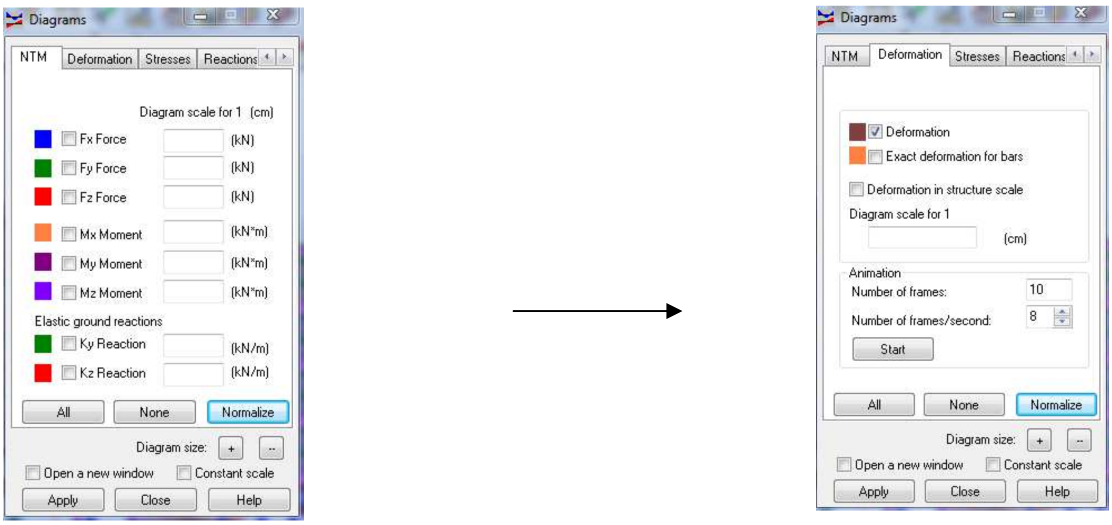


Figure III.59: Diagrams for bars dialog box.

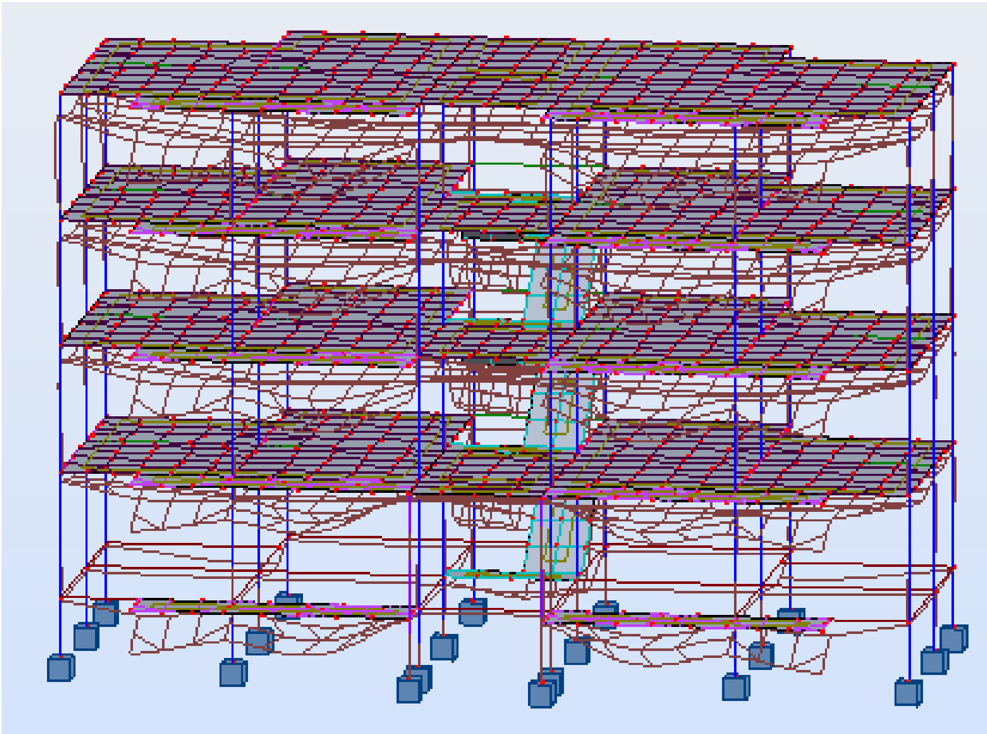


Figure III.60: 3D view of the structure deformation.

Bibliography

[1] Autodesk Robot Structral Analysis Professional 2010 - User guide.

[2] Autodesk Robot Structral Analysis Professional 2010 - New features of the software.

[3] Béton armée B.A.E.L 91 modifié 99 D.T.U associés (JEAN-PIERRE MOUGIN édition EYROLLES, 2000).

[4] Document technique réglementaire D.T.R BC 2 48. Règles parasismique algériennes RPA 99 /version 2003.

[5] Document technique réglementaire (D.T.R. C 2-4.7). Règlement neige et vent "R.N.V.1999".

[6] Document technique réglementaire (D.T.R. BC 2.2). Charges permanentes et charges d'exploitation.