Ref. ……………/2024

People's Democratic Republic of Algeria
Ministry of Higher Education and Scientific Research
Akli Mohand Oulhadj University - Bouira
Faculty of Science and Applied Sciences
Department of Civil Engineering

Handout

of practical work for

COMPUTER-AIDED DESIGN (CAD)

(in French: Calcul Assisté par Ordinateur, CAO)

In the field of bachelor's degree L3 in Civil Engineering

Semester: 6

Teaching unit: EMU 6.1

Directed by

Dr. AOUARI Issam

MCB : Department of Civil Engineering

April 2024

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.

Preface

This handout entitled COMPUTER-AIDED DESIGN (CAD) is intended for third-year LMD students in Civil Engineering. It focuses on the design of elements in a structure subjected to simple and compound solicitations. This handout is divided into three chapters.

The first chapter deals with the general presentation of the Etabs software. The main features of the software such as the calculation space, modeling, loading, and analysis of results are presented.

In the second chapter, four practical examples of Etabs Analysis software version 2020 are presented to understand the sizing mechanisms with this software. However, during the examples, some "modeling tricks" will be exposed to facilitate the user's approach to the multiple modeling choices offered by Etabs, and at the end of each example, exercises without solutions are given for the student to practice.

In the last chapter, the student becomes familiar with simple modeling concepts, normal force diagrams, shear forces, bending moments, deformation, dangerous section, stress, and finally design. This involves studying a project (Modeling a 4-storey reinforced concrete structure). Initially, we start with a complete presentation of the building, defining the various elements and the choice of materials used; this is followed by a second part which presents the modeling of structural elements (columns, beams, and walls), and non-structural elements (such as floors, balconies, and stairs). The third part will focus on the dynamic analysis of the structure, determining seismic action, and the structure's own dynamic characteristics during its analysis.

List of Abbreviations

CAD: Computer-aided design

ETABS : Extended Three-dimensional Analysis of Building Systems

FLAC : Fast Lagrangian Analysis of Continua

GC : Civil Engineering

MEF: Finite Element Method

OpenSees : Open System for Earthquake Engineering Simulation

SAP2000 : Structural Analysis Program

Seismosoft : Seismic analysis software

STAAD : Structural Analysis and Design Program

Tekla : Tekla Structures Software

TABLE OF CONTENTS

1. Introduction

INTRODUCTION

The process of designing, developing, and producing digital models of actual items or systems is revolutionized by computer-aided design (CAD) technology. It is now a crucial tool used in many other fields, such as engineering, manufacturing, product design, and architecture. With the use of CAD software, engineers and designers can easily and quickly produce elaborate, precise, and detailed designs.

Since the advent of computer-aided design (CAD), digital tools that provide increased productivity, precision, and flexibility have largely replaced manual drafting approaches in traditional design procedures. With CAD, designers may experiment with different design iterations, simulate real-world conditions, and work together with team members in different places with ease.

Key features of CAD software include:

- ✓ 3D Modeling: CAD makes it possible to create three-dimensional models of structures or things, giving designers the ability to view and work with designs in a virtual setting.
- ✓ Precision and Accuracy: accurate design representation is ensured by the use of CAD software, which makes exact measurements, alignments, and geometric calculations easier.
- ✓ Parametric Design: designers can more easily update and adapt designs when needs change when they use parametric modeling in CAD to build links between various design elements.
- ✓ Visualization and Rendering: before a thing is manufactured, designers and other stakeholders may see it thanks to the realistic visualization tools offered by CAD software.
- ✓ Collaboration and Documentation: team members may collaborate more easily by sharing designs, annotating drawings, and keeping track of revisions thanks to computer-aided design (CAD). Additionally, it simplifies the process of producing comprehensive documentation, such as bills of materials, schematics, and drawings.
- ✓ Integration with other Tools: the entire product development process can be streamlined by integrating CAD software with other software tools like computer-aided manufacturing (CAM) and computer-aided engineering (CAE).

Overall, CAD has become an indispensable tool for modern design and engineering, empowering designers and engineers to innovate, iterate, and bring their ideas to life more efficiently than ever before.

In this Handout, we present the roles of Computer-aided design (CAD) role in civil engineering field. CAD software revolutionizes civil engineering by empowering engineers to create detailed designs of infrastructure elements such as roads, bridges, buildings, and utilities with precision, ensuring compliance with engineering standards. Through 3D modeling, civil engineers gain a comprehensive view of projects, enhancing visualization and communication among stakeholders. With tools for terrain modeling and analysis, CAD facilitates the assessment of site topography and optimization of designs to accommodate environmental factors. Integration with GIS and GPS data enables engineers to incorporate geospatial information accurately. Parametric design capabilities allow for the creation of adaptable designs, while structural analysis tools ensure the integrity of infrastructure elements. Additionally, CAD streamlines quantity estimation, cost analysis, and construction documentation, promoting efficient project management. Collaboration features enable multidisciplinary teams to share designs, coordinate activities, and resolve conflicts effectively, thus enhancing project efficiency and success.

The purpose of CAD software is to employed the computational methods that are used in many modern computer programs for the seismic analysis of three-dimensional structural systems [1] . We can summarize the role of CAD in civil engineering field in the following point;

- 3D Modeling;
- Terrain Modeling and Analysis;
- Geospatial Integration;
- Parametric Design;
- Structural Analysis;
- Quantity Estimation and Cost Analysis;
- detailed drawings, and plans;
- Collaboration and coordination among multidisciplinary teams.

The handout is structured into many important chapters that are meant to give readers a thorough understanding of calculation software as it relates to civil engineering projects. By describing the importance of using CAD software in contemporary engineering procedures, the introduction sets the scene.

In CHAPTER 1, the basic ideas of computation software are covered, along with the methods and guidelines that are necessary for efficient use. This chapter, "Getting Started with an Available CAD Software," provides a helpful overview of how to use CAD tools, including basic operations, interface navigation, and vital functions. The use of ETABS software for the analysis and monitoring of actual civil engineering projects is then further explored in the study and monitoring of a real project using ETABS software. Case studies, real-world examples, and tips for using ETABS for structural analysis and design are all included in this second chapter.

In summary, the research highlights the importance of computer-aided design (CAD) software for improving productivity, accuracy, and efficiency in civil engineering projects. It also highlights future prospects and implications of the findings. In order to give engineers and researchers a thorough resource to help them utilize computation software in their work, the handout has adopted an organized approach.

CHAPTER 1:

BASIC CONCEPT ON CALCULATION SOFTWARE

CHAPTER 1: BASIC CONCEPT ON CALCULATION SOFTWARE

I.1 Objectives and Prerequisites of the chapter

At the end of this chapter, the student will be able to:

- ✓ Understand the mode of operation of calculation software in Civil Engineering.
- ✓ Know the difference between closed and open software.

the prerequisites of the student must know:

- ✓ Basic notions about drawing software (CAD) in Civil Engineering

I.2 Introduction

The world has been revolutionized by software in general. Any field you can think of has a number of software applications designed to make work easier and help us put our bright imaginations on paper. The same is true in the field of civil engineering, where a variety of highly developed computer software tools have produced quicker fixes and more precise outcomes.

I.3 History of CAD software

Since their creation, designers have utilized computers for computations. Even before the 1949 "Whirlwind" prototype, power system study and optimization made use of digital computers. Power network approach, or circuit design theory, was algebraic, symbolic, and frequently vector-based [2].

A number of advancements in computer software were produced between the mid-1940s and the mid-1950s. Servo-motors driven by generated pulses (1949), a digital computer with integrated functions to automatically coordinate transforms to compute vectors related to radar (1951), and the mathematical process of forming a shape graphically using a digital machine tool (1952) are a few of these innovations [3].

When Douglas T. Ross, an MIT researcher, observed radar operators using "interactive display equipment" in 1953. For his data reduction group tied to SAGE, he believed it would be perfect. When debugging software, programmers could utilize logical switches that could be opened and closed and flowcharts on a display scope thanks to utility apps made by the makers of these antique systems. They discovered that they could use geometric forms and electronic symbols to create simple flowcharts and circuit diagrams. [4].

Other notable occurrences in the 1960s and 1970s included the application in 1969, the development of CAD systems United Computing, Intergraph, IBM, and Intergraph IGDS in 1974 (which led to the creation of Bentley Systems MicroStation in 1984), and the introduction of commercial CAD systems in the 1970s from Japanese manufacturers Seiko and Zuken.

In the late 1980s and early 1990s, ShapeData's Parasolid, ACIS (Spatial Technology Inc.), and B-rep solid modeling kernels engines for manipulating geometrically and topologically coherent 3D objects were developed. These developments were also crucial to the development of CAD.

I.4 Software in Civil Engineering

Numerous software packages are available for each discipline in civil engineering, such as geotechnics, structures, materials, transport, hydraulics, environmental engineering, project management. We can distinguish between two types of software in civil engineering; drawing software and calculation software. for calculation software, we can mention: SAP2000, ETABS, ROBOT, TEKLA, REVIT, CIVIL3D, STAAD and PLAXIS. And we can mention drawing software: Autocad, Archicad, Mapinfo, 3DsMax, Home 3D, …

Figure 1.1: the most used civil engineering software

I.5 Mode of operation of calculation software

The calculation software in Civil Engineering makes it possible to model, design, analyze, verify and display the geometry of the structure, as well as the results of the analysis. The analysis procedure can be divided into three phases [1]:

1) Input data

The choice of the type of modeling, the software as well as the modeling details are directly correlated to the input data and the desired objectives. In this phase, we must specify the nature, the origin of the input data as :

- Geometry;
- Material;
- Charging mode.

2) Processor

This phase is very important, where, the software uses computational methods to solve the problem, and find the desired results.

3) Post-processing

The object of this phase is the graphical display and printing of the results offered by the software, namely the graphical visualization and the recording of certain calculation results.

I.6 Calculation methods

The more complex the equations given by modeling have become, the more difficult it has become to solve them using perfect variables to obtain analytical solutions. To overcome this challenge and find a solution, the main approaches used are the following:

- Simplify physics to the maximum;
- Divide the model into smaller and simpler models;
- Use imperfect numerical algorithms to get as close as possible to the real solution.

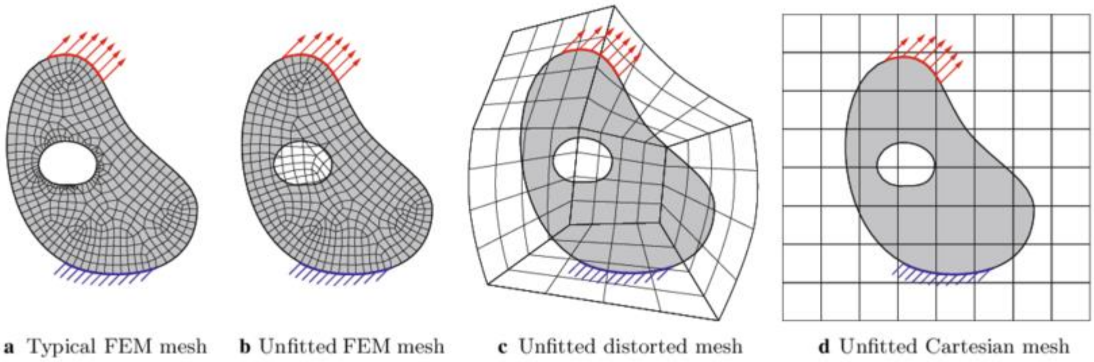
Mathematicians and physicists have found several numerical methods for solving these complex equations. The four most used methods are:

- The finite element method (FEM*)
- The finite volume method (FVM)
- The finite difference method (MDF)
- The boundary element method (BEM)

The finite element method (FEM) is the most recognized among these methods.

I.7 Finite element method

MEF is a numerical method that uses discretization to transform a continuous domain into a discrete domain. We calculate the equations for each of these areas and once that's done, we have to put them all together so that the solution makes global sense using the superposition theory.

(1) 

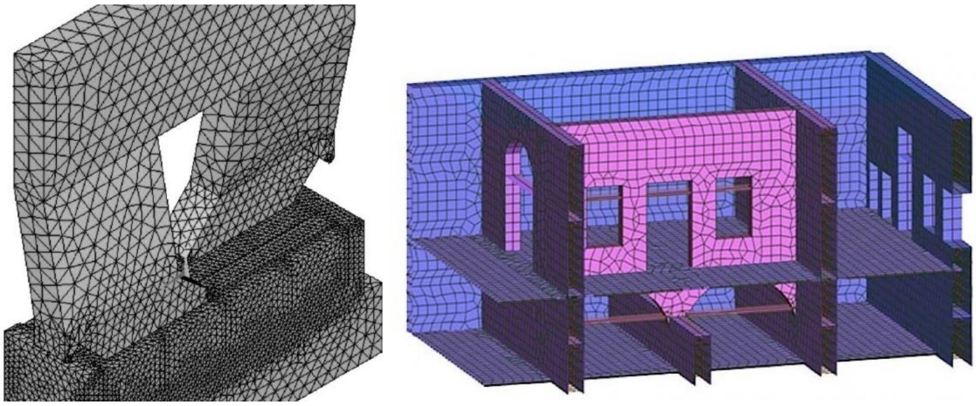
(2) 

Figure 1.2: Discretization by MEF

I.8 Open source and paid software

Open software: Open software means software using open standards or formats. The legal definition of an "open" standard or format is as follows: "open software offers interoperable data formats and whose technical specifications are public and without restrictions on access or implementation" as opposed to a closed or proprietary format. Open formats are interesting for their interoperability.

Exemple: *OpenSees, Cast3m, ...etc*

Paid software (or non-free software): Non-free software is any software that prevents users from simultaneously exercising the four software freedoms—executing the program for any kind of use, studying its source code and gaining access to it, distributing copies, and modifying and improving the source code—legally, technically, or through any other means.

Exemple: *SAP2000, ETABS, ROBOT, ...etc*

I.9 Advantages and limitations of software

a) **Advantages:** Time and / or budget constraints do not always make it possible to carry out tests to study the real behavior of the structures that we would like to test. With a simulation software, the calculations are precise and can take into account many parameters for a very thorough study of our designs. With a "simple" computer and a numerical simulation software, you are able to test an almost infinite number of parameters and to study a project that will not take any delay in realization.

b) **Disadvantages:** If there are certain advantages to using a CAD tool, we must not hide the disadvantages that result from it and that users report. Let us cite, for example:

- Most of these softwares use numerical methods such as MEF, or, we cannot always get a precise solution.
- The user must have theoretical knowledge to avoid errors in the results.
- Most of these softwares require long training sessions for an engineer to learn how to use them effectively.
- Sometimes software does not behave as expected, which forces the engineer to spend a lot of time solving problems instead of designing.

CHAPTER 2:

GETTING STARTED WITH AN AVAILABLE SOFTWARE

CHAPTER 2: GETTING STARTED WITH AN AVAILABLE SOFTWARE

II.1 Introduction

The introduction of this chapter aims to provide an overview of the ETABS software and to present its importance in the field of structural modeling. This guide focuses on the practical use of the software and aims to provide users with the knowledge and skills necessary to master ETABS effectively.

The objectives of this getting started guide are to help users acquire a thorough understanding of the use of the ETABS software and to support them in their learning. This guide provides step-by-step instructions, practical tips and concrete examples to allow users to quickly familiarize themselves with the essential features and techniques of ETABS. The ultimate goal is to empower users so that they can use the software effectively and efficiently for modeling and analyzing complex structures.

In this section a detailed presentation of the ETABS software, highlighting its key features and its use in the analysis and design of structures. Users will be informed about the power of this software, its advanced modeling capabilities and its integration with other engineering tools. Examples of real projects that use ETABS will also be presented to illustrate its usefulness in different fields.

II.2 Presentation of the ETABS software

For the analysis and design of multi-story buildings, ETABS is an engineering software program distinguish to structures analysis. This class of structure's distinct grid-like shape is coordinated with modeling tools and templates, code-based load prescriptions, analytical approaches, and solution strategies. ETABS can be used to analyze simple or complex systems in static or dynamic environments. P-Delta and Large Displacement impacts may combine with modal and direct-integration time-history analysis for a more complex evaluation of seismic performance. Under monotonic or hysteretic behavior, nonlinear connections, concentrated PMM, or fiber hinges can capture material nonlinearity. It is feasible to develop applications of any complexity thanks to intuitive and integrated features. For designs ranging from straightforward 2D frames to intricate modern high-rises, ETABS is a coordinated and effective tool because to its interoperability with a number of design and documentation platforms.

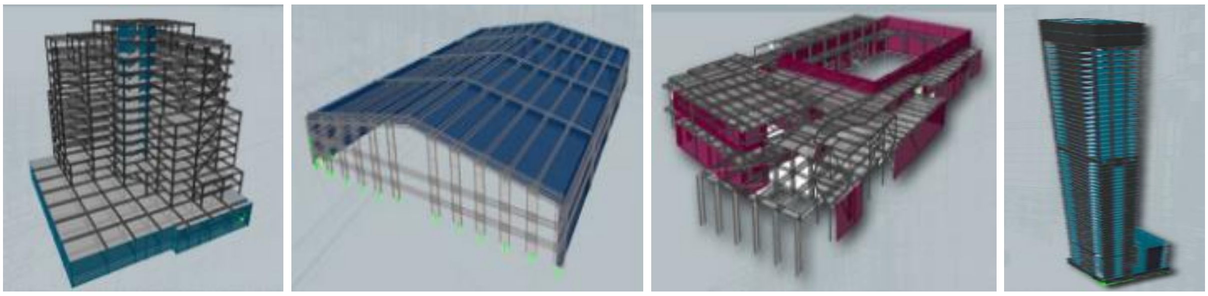


Figure 2.1: Modeling of Structural Systems in Etabs

Developers Computers and Structures (CSI) created the popular structural analysis and design program ETABS (Extended Three-dimensional Analysis of Building Systems). It is renowned for its advanced capabilities in the analysis, design, and detailing of building structures. Here's an overview of the features and functionalities of ETABS:

- ***Modeling Capabilities:***
  - ETABS offers a user-friendly graphical interface for creating and editing structural models.
  - It supports various types of structural elements such as beams, columns, slabs, walls, shells, and joints.
  - Users can create models using both physical and conceptual representations, allowing for efficient modeling of complex structures.
- ***Analysis Options:***
  - ETABS provides a wide range of analysis options including static, modal, response spectrum, time history, and nonlinear analyses.
  - It employs sophisticated algorithms to accurately predict the behavior of structures under different loading conditions.
- ***Design Codes:***
  - The software supports numerous international building codes and design standards, enabling engineers to design structures compliant with local regulations.
  - Design codes include those for steel, concrete, composite, and timber structures.
- ***Integrated Design:***
  - ETABS facilitates integrated design by performing analysis and design within the same environment.

- It automatically checks designs for compliance with relevant design codes and provides detailed reports of design results.

➢ ***Advanced Analysis Techniques:***

- ETABS offers advanced analysis techniques such as pushover analysis for assessing structural performance under lateral loads and nonlinear static analysis for predicting behavior beyond linear range.
- Time history analysis allows engineers to simulate the dynamic response of structures to earthquake or other transient loads.

➢ ***Result Visualization:***

- The software provides comprehensive visualization tools for displaying analysis results, including deformations, stresses, forces, and mode shapes.
- Users can generate detailed graphical and tabular reports for further analysis and documentation purposes.

➢ ***Interoperability:***

- ETABS supports interoperability with other software applications through various file formats such as DXF, DWG, IFC, and CIS/2.
- It also integrates seamlessly with other CSI software products like SAP2000 for advanced analysis and SAFE for foundation design.

➢ ***User Support and Training:***

- CSI offers extensive user support including technical documentation, tutorials, online forums, and direct customer support.
- Training programs are available for users to enhance their proficiency in utilizing the software effectively.

ETABS is widely used by structural engineers and designers for the analysis, design, and optimization of buildings and other structures. Its comprehensive features, user-friendly interface, and robust analysis capabilities make it a preferred choice for projects ranging from simple structures to complex high-rise buildings and industrial facilities.

II.3 ETABS software different version

As of last update in January 2022, the ETABS software has undergone several version releases, each introducing new features, enhancements, and bug fixes. While I can't provide specific details on versions released after that date, I can give you an overview of some of the versions available up to that point:

✓ **ETABS 9.x Series:**

The 9.x series marked significant advancements in modeling capabilities, analysis methods, and design tools. It introduced features such as improved graphical user interface (GUI), enhanced modeling options for various structural elements, and expanded analysis capabilities.

Figure 2.2: interface of Etabs 9 series

✓ **ETABS 2013:**

ETABS 2013 introduced several new features and enhancements including faster performance, improved modeling tools, and enhanced analysis capabilities. It also included improvements in design code support and interoperability with other software applications.

Figure 2.3: interface of Etabs 2013 version

✓ **ETABS 2015:**

ETABS 2015 brought further enhancements to modeling, analysis, and design capabilities. It introduced features like cloud-based licensing, more advanced nonlinear analysis options, and improved integration with other CSI software products.

Figure 2.4: interface of Etabs 2015 version

✓ **ETABS 2016:**

ETABS 2016 continued to improve upon the features introduced in previous versions. It included enhancements in analysis and design, as well as improvements in performance and stability.

Figure 2.5: interface of Etabs 2016 version.

✓ **ETABS 2017:**

ETABS 2017 focused on improving the user experience with enhancements to the graphical interface and workflow. It also introduced new features such as the direct analysis method (DAM) for concrete frame design and enhancements to nonlinear analysis capabilities.

Figure 2.6: interface of Etabs 2017 version

✓ **ETABS 2018:**

ETABS 2018 introduced significant improvements in modeling, analysis, and design capabilities. It included features such as code-based wind load generation, performance-based design options, and enhancements to the API for customization and automation.

Figure 2.7: interface of Etabs 2018 version

✓ **ETABS 2019:**

ETABS 2019 brought further improvements in performance, stability, and user experience. It included enhancements to the graphical interface, modeling tools, and analysis capabilities, as well as improvements in design code support.

Figure 2.8: interface of Etabs 2019 version

✓ **ETABS 2020:**

ETABS 2020 continued to enhance the software's capabilities with improvements in modeling, analysis, and design. It introduced features like data visualization tools, enhanced interoperability with other software applications, and improvements in performance-based design options.

Figure 2.9: interface of Etabs 2020 version.

✓ **ETABS 2023:**

In ETABS 2023, several enhancements have been implemented to improve the software's functionality. The key changes are:

- *Analysis Log Form*: When the analysis log form is displayed, it will always remain in front of the main graphical user interface. Additionally, when the main graphical user interface is minimized or restored, the analysis log form will follow suit.
- *Analysis Messages*: Enhancements have been made to the analysis messages, including providing the affected element type and element name when applicable, displaying Windows system error message text for file I/O errors, and omitting "Results deleted" informational messages for certain load case operations.
- *Cracked Section Analysis:* The cracked section analysis options include two convergence check options: SRSS of Vertical Displacements (same as in ETABS v20) and Max Absolute Vertical Displacement (newly added). The latter is useful for single-story models where deformation primarily occurs in objects with cracked-section properties.
- *Analysis Monitor Form:* The Analysis Monitor form displays the run tag and status of completed load cases, making it easier to identify load cases that didn't complete during or after analysis

Figure 2.10: interface of Etabs 2023 version

II.4 The Window for ETABS

Figure II.10 depicts the ETABS graphical user interface, which consists of the main window, status bar, mouse pointer location coordinates, model explorer, display windows, menu bar, toolbars, and current units. The bulleted list that follows has descriptions for each of these objects.

Main Window: Standard Windows functions can be used to move, resize, maximize, minimize, or dismiss this window. For more details on those items, see Windows help, which may be accessed through the Start menu.

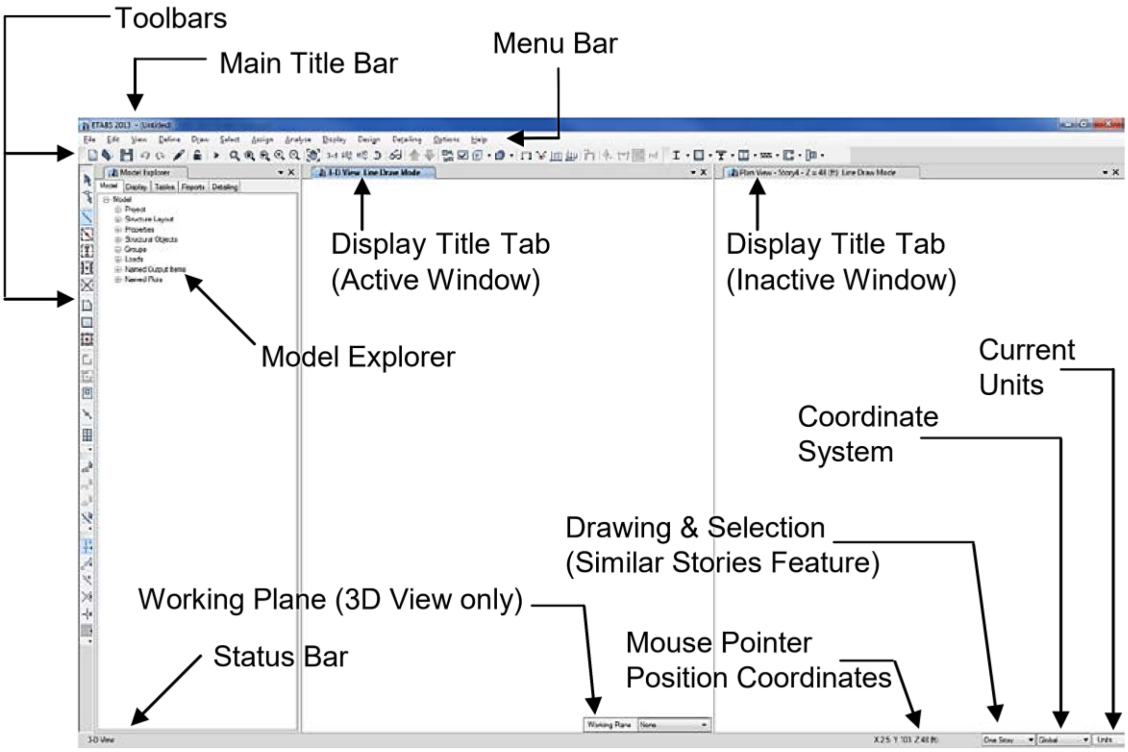


Figure 2.11: The ETABS graphical user interface

Main Title Bar: The program and model names are displayed in the main title bar. When the application is running, the main title bar is highlighted. To move the main window around the computer screen, hold down the mouse button as you left-click in the main title bar and drag the window.

Menu Bar: The program's menus are displayed on the menu bar, where users can choose different commands to carry out different tasks.

Buttons and Toolbars: Buttons make up toolbars. Commands that are often used can be accessed "single-click" via buttons. A brief description of a toolbar button's function will appear in a small text box when you hover the mouse pointer over it for a few seconds without clicking or holding down any mouse buttons.

Model Explorer: Using a hierarchical tree structure, the model explorer provides quick access to model definition data, including property forms, load definitions, and object forms. It also facilitates the analysis, design, and description of results in tabular, report, and graphical formats. The Model Explorer groups these items under five tabs: Model, Display, Tables, Reports, and Detailing. A right-click on a "leaf" in a tree will display a context-sensitive menu (things indicated in bold in the menu indicate the default action that would occur if the user double clicks on the leaf). Trees can be expanded by clicking on a node. To assign a section to a model, just drag the section from the tree onto the relevant model object (for example, a frame section onto a frame object). Modular changes can be accelerated greatly using this drag-and-drop method.

Display Windows: In addition to displaying the model's geometry, a display window may additionally incorporate loading, property displays, analysis or design outcomes, and details. The quantity of windows that can be shown is unlimited.

Tab for Display Title: The tab for display title is situated at the top of the window for display. When the related display window is open, the tab with the title of the display is highlighted. Included in the text of the display title tab are usually the view type and location in the corresponding display window.

The status bar: is positioned at the base of the primary window. There is text on the left side of the status bar that describes the program's current state.

Working Plane Drop-Down List: When a drawing command is active, this drop-down list shows up in a 3-D View display window. Unless you use snaps, you can only draw objects in a 3-D view on the narrative (working plane) that you select from this drop-down list.

Mouse Pointer location Coordinates: The status bar's right side displays the mouse pointer location coordinates. The coordinates of the mouse pointer position can be shown without a window being active. All that needs to happen is for the mouse pointer to be over the window.

Drawing and Choosing List with a Drop-Down: The status bar's right side has this list with a drop-down. The drop-down list has three options: One Story, All Stories, and Similar Stories. One Story limits the creation of an object to the story level at which it is drawn. When an object is designed with All Stories, it generates objects in the model at every story level at the same plan location. Any object picked during an object select using All Stories causes all other objects at the same plan position to be selected at every story level. When using Similar

Stories, drawing an object causes all other objects in the same plan location to be selected at similar story levels, and selecting an object causes all other objects in the same plan location to be created at all comparable story levels in the model.

Coordinate System drop-down list: The Global Coordinate System and user-defined Grid Systems can be switched between using this list, which is located on the right side of the status bar. The mouse pointer location coordinates and the model's orientation are both impacted by the chosen system.

Units Current: The units currently in use are shown in a pop-up list that appears on the status bar's far right side. Any moment along the model-creation process, these units might be modified.

File operations: These include opening a new model, importing an already-existing model for editing or display, exporting or saving the active model for use in ETABS or another program, and generating output. The File menu is used to select file operations. One can begin creating new models from scratch or by using the program's pre-made templates.

Edit: Model modifications are accomplished by editing. The majority of editing actions operate on one or more objects that were chosen right before the Edit command was sent. With edit commands, objects can be extruded, merged, aligned, copied, pasted, relocated, and replicated.

View: Each display window may have different view settings, which impact how the structure appears, configured for it.

Define: This function is used to construct named entities outside of the model's geometry. These entities can be accessible through the Define menu and consist of load patterns, cases, combinations, and material features as well as frame, tendon, wall, and slab sections. Some of those entities can be defined using the Assign menu during the assignment operation, and defining them does not require selecting an object beforehand.

Draw: Drawing is the process of adding new objects to the model or changing one at a time. The items consist of various joint, frame, and shell objects as well as beams, columns, slabs, decks, walls, connections, and tendons. The application has to be in Draw Mode in order to draw. You can do this by using a Draw menu command or by clicking one of the draw buttons on the toolbar.

When in Draw Mode, you can draw and modify objects with the left mouse button and query their properties with the right mouse button. A "Properties of Object" form that can be used to indicate different structural properties and, in the case of numerous towers, the tower to which the object belongs appears depending on the type of object to be depicted. Frame properties can be set at the same time as frame objects are drawn. When drawn, shell objects can be designated as apertures or given wall or floor features. An item drawn can be selected, loads assigned to it, or current assignments changed once the object has been drawn.

Select: Selection is the process of determining which objects will be subject to the subsequent operation.

Assign: When drawing an object, you may assign certain properties to it. For example, you may assign a structural property to a frame object. However, by using the Assign menu command, one or more objects that were selected right before may have new assignments made to them, or assignments already made may be changed. Properties, constraints, loads, and group names are examples of assignment operations.

Analyze: The displacements, forces/stresses, and reactions that occur from utilizing the previous commands to generate a comprehensive structural model can be found by analyzing the model. Prior to doing an analysis, make sure that the objects are connected and do not overlap by using the Check Model command and the Set Active Degrees of Freedom command on the Analyze menu.

When conducting an analysis for the first time, choose which cases to run by selecting Set Load Cases to Run from the Analyze menu. After selecting the load instances, you can start the analysis by clicking the start Analysis button on the toolbar or by selecting Run Analysis from the Analyze menu. It is not necessary to run any cases that have already been run again. If a load case is selected that depends on the outcome of another case, the prerequisite case will execute first, if it hasn't already.

Display: The model and the analysis's findings can be viewed using the Display menu commands. This application has tabular and graphic displays accessible. Toolbar buttons can be used to access display items, or the Display menu can be used to select them.

Graphical Displays: For every display window, a variety of graphical display options are available. A window's view orientation and display settings may also differ from one another.

It is possible to display loads, analysis results, and undeformed geometry. You can click on an object with the right mouse button to see more details about the results that are displayed.

Tabular Displays: By selecting the Tables tab in the Model Explorer, tabular data for the model can be shown. Select a table to view, then use the right-click menu. Certain tables will only be available for the objects that have been selected before using the commands.

Design: Following the completion of an analysis, the design code requirements can be followed when designing frames, composite beams and columns, joints, shear walls, slabs, and steel connections. Selecting the relevant Design menu command will allow you to perform design for the provided design combinations. Make sure the chosen design codes and preferences are correct before you start creating by using the relevant View/Revise Preferences command from the design menus.

Detailing: You can manage how schematic construction documents are arranged and laid out using the Detailing option. Specifications may include things like drawing size and layout, section cuts, schedules for columns, beam framing plans, shear wall reinforcement, layouts for composite slab reinforcing, general notes, cover sheets, and more. Usually, after analysis and design are finished, this menu is accessible. By choosing the Detailing tab in the Model Explorer, you can also see the created drawing sheets and views.

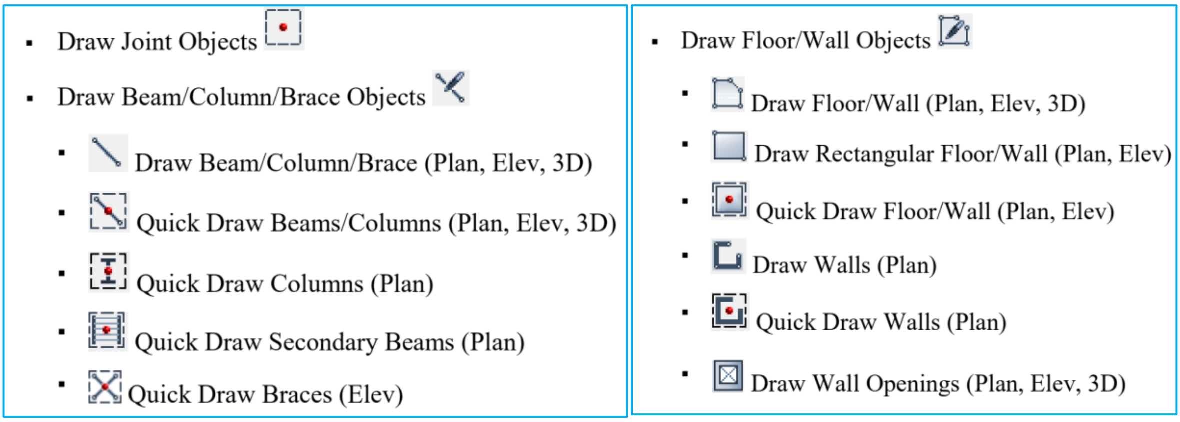


Figure 2.12: example of draw tools in Etabs

II.5 Begin a Model with Etabs

In this section, we go over how to set up the fundamental grid system in order to start a model. The positioning of structural elements is in relation to the grid system. [5].

To start a new model, you can click the **File menu** > **New Model** command.

On the Start Page, click the New Model button to see the Model Initialization form shown in the following figure.

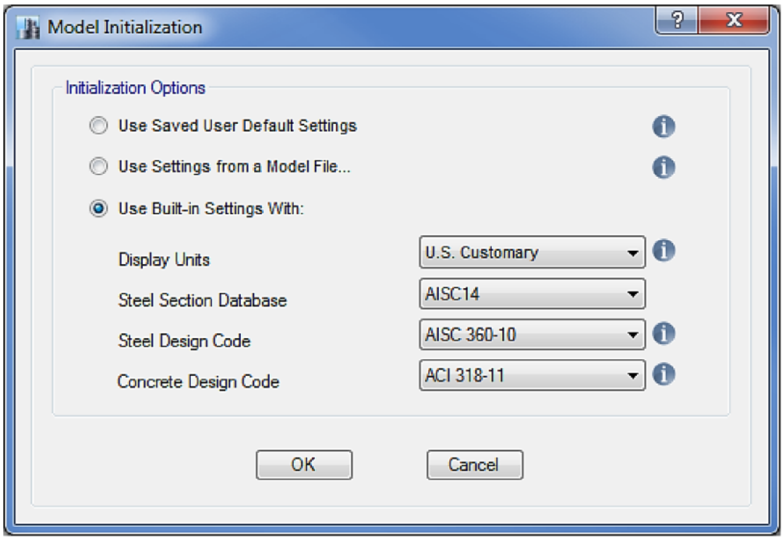


Figure 2.13: Model Initialization form

Select the Use Built-in Settings With option on the Model Initialization form. Next, select U.S. Customary, Metric SI, or Metric MKS from the Display Units drop-down box to configure the input and display unit defaults.

The New Model Quick Templates form, seen in Figure 2-14, will appear once you click the OK button on the Model Initialization form. The horizontal grid line spacing, story data, and template data are specified using the New Model Quick Templates form. Starting a model is quick and simple with template models. They populate the model automatically with structural objects that have the right attributes. We strongly advise you to use templates to begin building your models whenever you can.

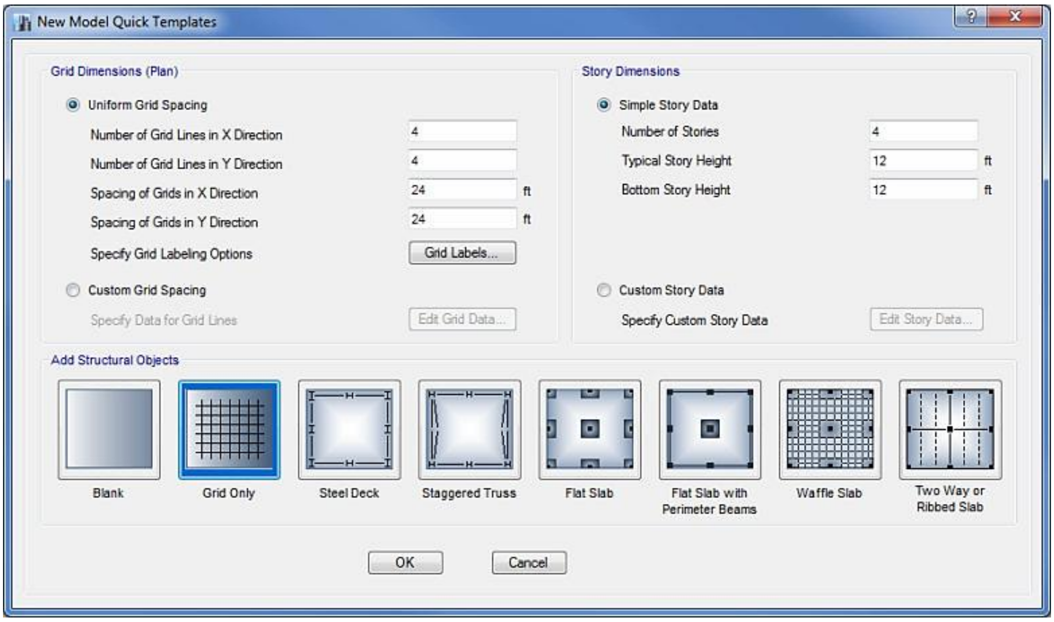


Figure 2.14: New Model Quick Templates form

To specify a grid line system, use the Grid Dimensions (Plan) section of the form. To define the grid line system, choose one of the following two options:

Uniform Grid Spacing: The regular spacing between the grid lines in the X and Y directions should be specified, along with their number. Keep in mind that there may be variations in the consistent spacing in the X and Y directions. This setting establishes a grid system exclusively for the global coordinate system.

Custom Grid Spacing: For the global coordinate system, provide X and Y directions with nonuniformly spaced grid lines. To make changes to the grid system after selecting this option, click the Edit Grid Data button.

Define Story Data: To specify the quantity and height of stories, use the form's Story Dimensions section. You have two options to establish the story data: either by Custom Story Data (after selecting this option, click the Edit narrative Data button to open the Story Data form) or by Simple Story Data (enter values in the edit boxes to determine the number of stories and a typical story height).

II.6 Define Properties

The sections that follow provide instructions on how to evaluate program defaults and define new properties. [5].

a) Material Properties

The Define Materials form, illustrated in Figure 2.15, will appear when you select the **Define menu** > **Material Properties** command.

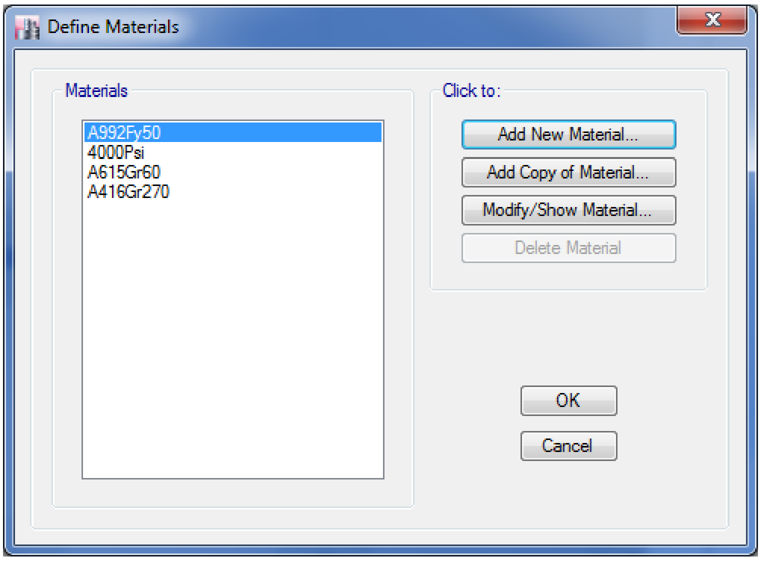


Figure 2.15: Define Materials form

Both the definition of new properties and the review of current materials are possible with the Define Materials form. Click the Add New Material button on the Define Materials form to add a new material. Choose a material from the Material Type drop-down list, followed by a Standard and Grade from the corresponding drop-down lists, when the Add New Material Property form displays as seen in Figure 2.16.

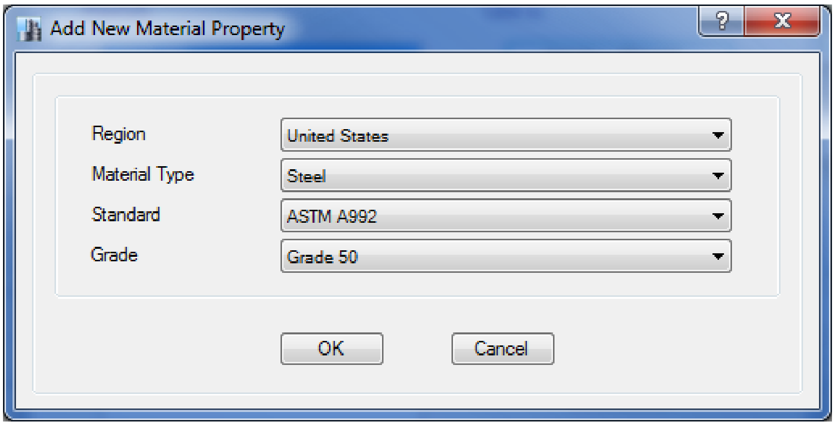


Figure 2.16: Figure showing the process of adding a new material property.

After making your choices on the Add New Material Property form, click the OK button to bring up the Material Property Data form, where you may check and amend the new material's data.

The user has the ability to modify the default values that the software displays for these attributes, specifically for the two materials, steel and concrete. As the following table illustrates, different types of materials can be modified by inputting the mechanical properties in the form that appears on the screen.

Table 2.1: Mechanical characteristics for the calculation of the stresses.

Terms in Etabs	Default values
Mass per unit volume	2548,538 kg/m^3
Weight per unit volume	24,9926 kN/m^3
Modulus of elasticity	31000 MPa
Poisson's ratio	0.2
Coeff of thermal expansion	0.00001 1/C
Shear Modulus	12916.67 MPa
Concrete strength, fc	25 MPa
Reinforcing yield stress, fy	440 MPa
Shear steel yield stress, fys	484 MPa
Minimum tensile strenght, fu	550 MPa

b) Frame Sections

To view the Frame Properties form, click the Define menu > Section Properties > Frame Sections command. Click the Import New Properties button to enable steel frame sections from property files, or click the Add New Property button to add user-defined sections. Both options will bring up the Frame Property Shape Type form, as seen in Figure 2.17.

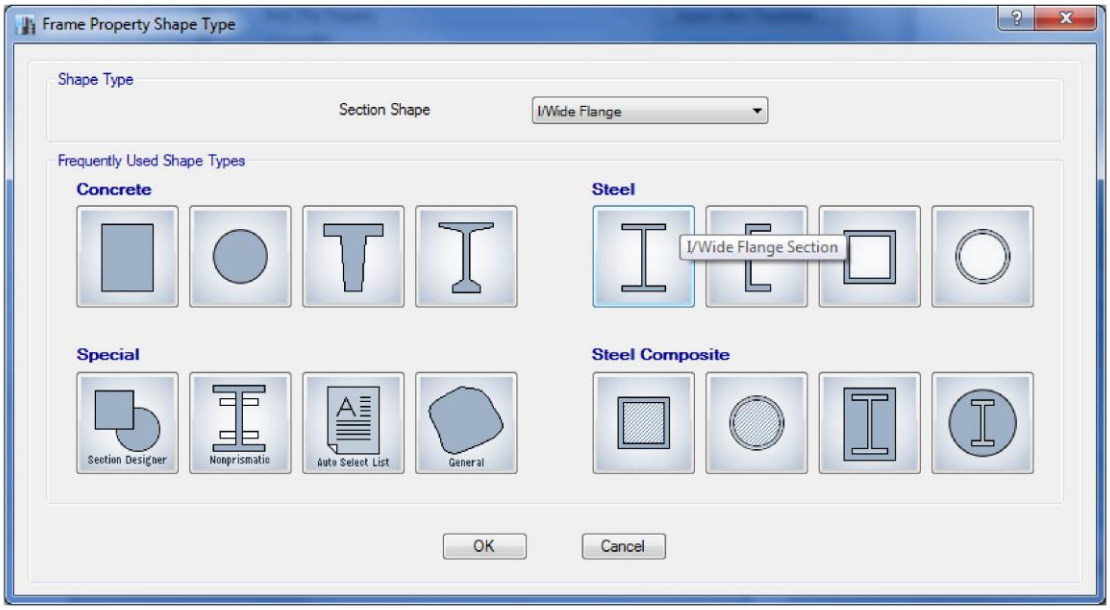


Figure 2.17: Frame Property Shape Type form.

II.7 Draw structural objects

In addition to drawing items like floors, walls, beams, and columns manually, as shown in the next sections, it can also construct a model using a template.

a) Create columns

You can use the **Draw menu** > **Draw Beam/Column/Brace Objects** > **Quick Draw Columns** command, or you can click the **Quick Draw Columns** button. For the columns in Figure 2.18, the Properties of Object box will appear docked in the lower left corner of the screen.

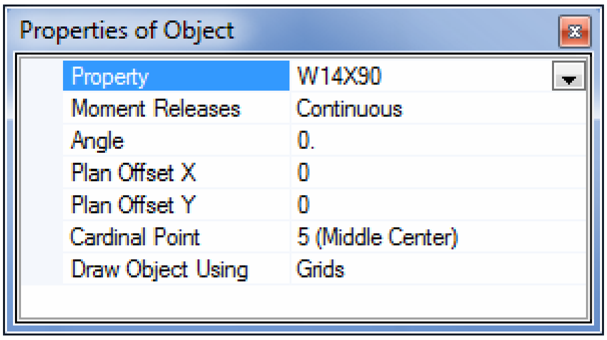


Figure 2.18: Properties of Object Box for Columns

Once the parameters have been verified in the Properties of Object box, left-click once in the Plan View at the grid line intersection where the column is to be placed. At that point in the Plan View, an I-shaped column should show up. Place the other columns in the same way.

If you move the mouse over the grid line intersections in the model, a selection box that looks like the one in Figure 2.19 should enlarge. The software will draw the column objects at the intersections of the grid lines inside the selection box bounds when the left mouse button is released.

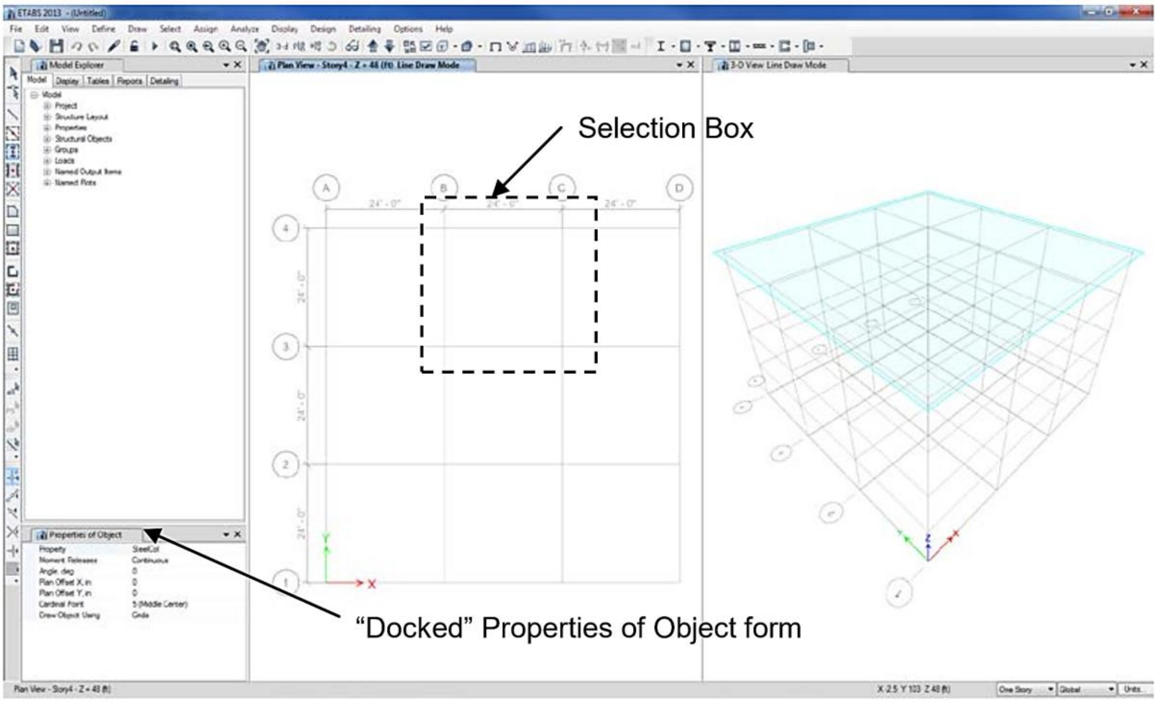


Figure 2.19: Creating objects in columns within a windowed area.

b) Draw beams  

Either select Quick Draw Beams/Columns from the Draw menu or Quick Draw Beams/Columns from the Draw menu. Docked in the lower left corner will be the Properties of Object box for beams as seen in Figure 2.20.

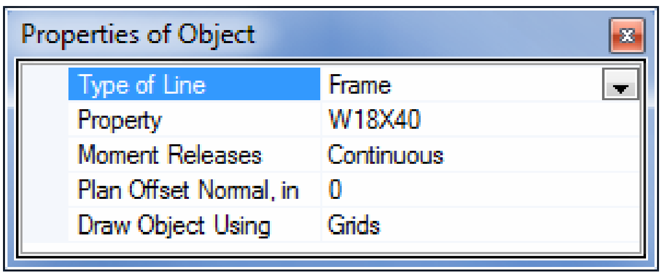


Figure 2.20: Properties of Object Box for Beams

c) Create the secondary beams 

By selecting the Quick Draw Secondary Beams button or the Draw menu > Draw Beam/Column/Brace Objects > Quick Draw Secondary Beams command, secondary or "infill" beams can be added. As with the other drawing procedures, you will see a Properties of Object box docked in the lower left corner where you may specify the secondary beam characteristics.

d) Draw the Floor 

To draw a floor or wall, either click the Draw button or use the Draw menu > Draw Floor/Wall Objects > Draw Floor/Wall command. The docked Properties of Object box will show up in the lower left corner for the locations depicted in Figure 2.21.

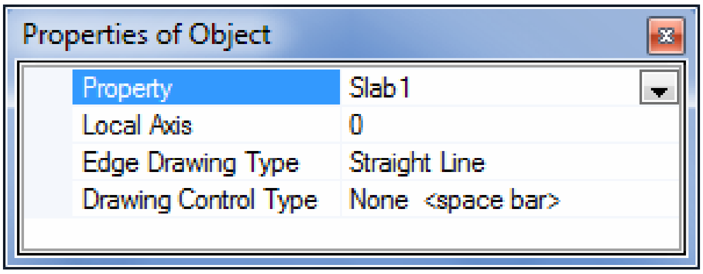


Figure 2.21: Properties of Object Box for Shells

e) Draw Walls 

To create walls, either click the Draw Walls button or choose the Draw menu > Draw Floor/Wall Objects > Draw Walls command. If you select the walls property box in Figure 2.22, the docked version of the box will appear in the lower left corner.

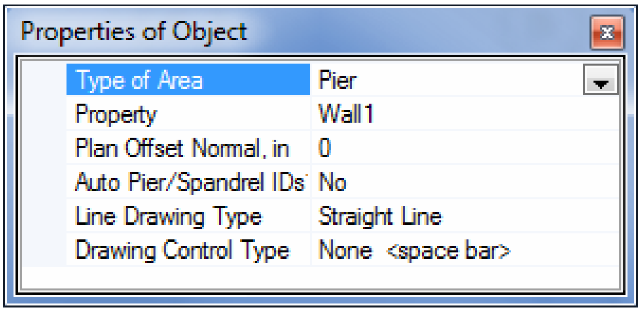


Figure 2.22: Properties of Object Box for Walls

f) Draw Wall Stacks 

Select the command "Draw Wall Stacks" from the Draw menu or click the Draw Wall Stacks button. This will bring up the New Wall Stack form that you see in Figure 2.23.

Click on the representative symbol to select any of the pre-defined wall stacks. Changes can be made to the wall segments' lengths and thicknesses by entering new values in the edit boxes shown on the Layout Data tab. Click OK after you've checked all the wall stack parameters on the Layout Data tab.

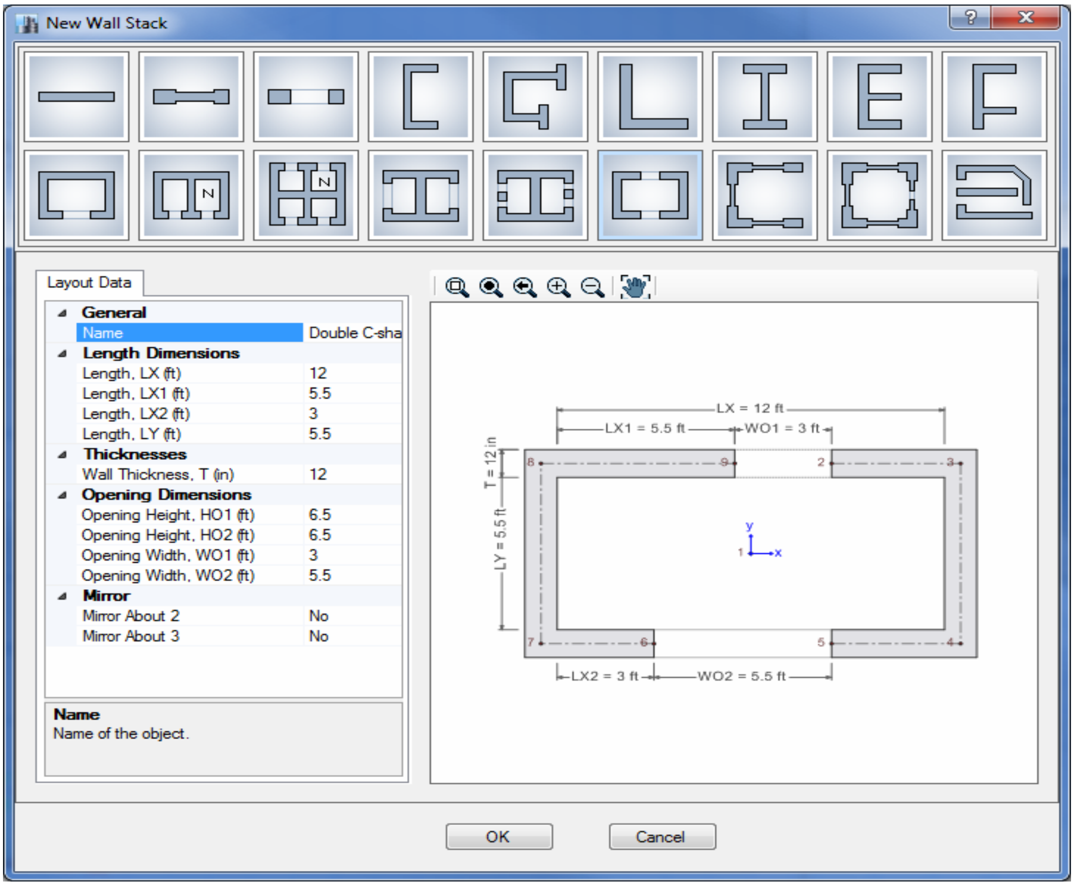


Figure 2.23: New Wall Stack form.

II.8 Assign/Restraints

The supports can be specified as articulated, recessed, or as recessed with certain relaxations.

The articulated support is considered to be released in rotation, and blocked in translation. ETABS also makes it possible to specify spring constants, in translation or rotation, which allows the definition of elastic supports.

The user of the program must specify in a certain number of nodes, the ways of fixing the structure with the external environment (support), as well as between these elements. In general, the connection of two elements in a node can be an articulation, an embedding or a blocking of a few degrees of freedom. In ETABS, all nodes are recognized as rigid nodes by default.

Figure 2.24: Different types of supports.

II.9 Assign/Change Properties

The user draws joint, frame, shell, link, and tendon components when building the model. Those items need to be given properties, like material properties, frame sections, wall/slab/deck sections, link properties, tendon properties, and loads, among others, in order to facilitate analysis and design.

The available assignment types are dependent on the type of object, as Table 2.2 illustrates. The type of design (steel versus concrete versus composite design, for example) affects assignments as well.

Table 2.2: Possible Assignments to Objects by Object Type

Object	Assignment Option	Name of Input Form*
Joint		Joint Assignment -
	Restraints	Restraints
	Springs	Springs
	Diaphragms	Diaphragms
	Panel Zone	Panel Zone Property
	Additional Mass	Additional Mass
	Joint Floor Meshing Options	Joint Floor Meshing Option
Frame		Frame Assignment -
	Section Property	Section Property
	Property Modifiers	Property Modifiers
	Releases/Partial Fixity	Releases/Partial Fixity
	End Length Offsets	End Length Offsets
	Insertion Point	Insertion Point
	Local Axes	Local Axes
	Output Stations	Output Stations
	Tension/Compression Limits	Tension/Compression Limits
	Hinges	Hinges
	Hinge Overwrites	Hinge Overwrites
	Line Springs	Line Springs
	Additional Mass	Additional Mass
	Pier Label	Pier Label
	Spandrel Label	Spandrel Label
	Frame Auto Mesh Options	Frame Auto Mesh Options
	Frame Floor Meshing Options	Frame Floor Meshing Option
	Moment Frame Beam Connection Type	Moment Frame Beam Connection Type
	Column Splice Overwrite	Column Splice Overwrite
	Nonprismatic Property Parameters	Nonprismatic Property Parameters
	Material Overwrite (not applicable to section designer, nonprismatic, auto select, encased rectangle/circle, or filled tube/pipe sections)	Material Overwrite
Shell		Shell Assignment -
	Slab Section	Slab Section
	Deck Section	Deck Section
	Wall Section	Wall Section
	Openings	Openings
	Stiffness Modifiers	Stiffness Modifiers
	Thickness Overwrites	Thickness Overwrites

II.10 Load the Structural Model

ETABS automatically produces and assigns code-based loading conditions for gravity, seismic, wind, and thermal forces after modeling is finished. There is no limit to the number of load situations and combinations that users can specify.

Then, advanced nonlinear approaches for characterizing static-pushover and dynamic response are provided by analysis capabilities. Modal, response-spectrum, and time-history analysis are examples of dynamic considerations. Geometric nonlinearity is explained by the P-delta effect.

develop features will automatically scale elements and systems, develop reinforcing schemes, and otherwise optimize the structure based on desired performance metrics given the encompassing specification.

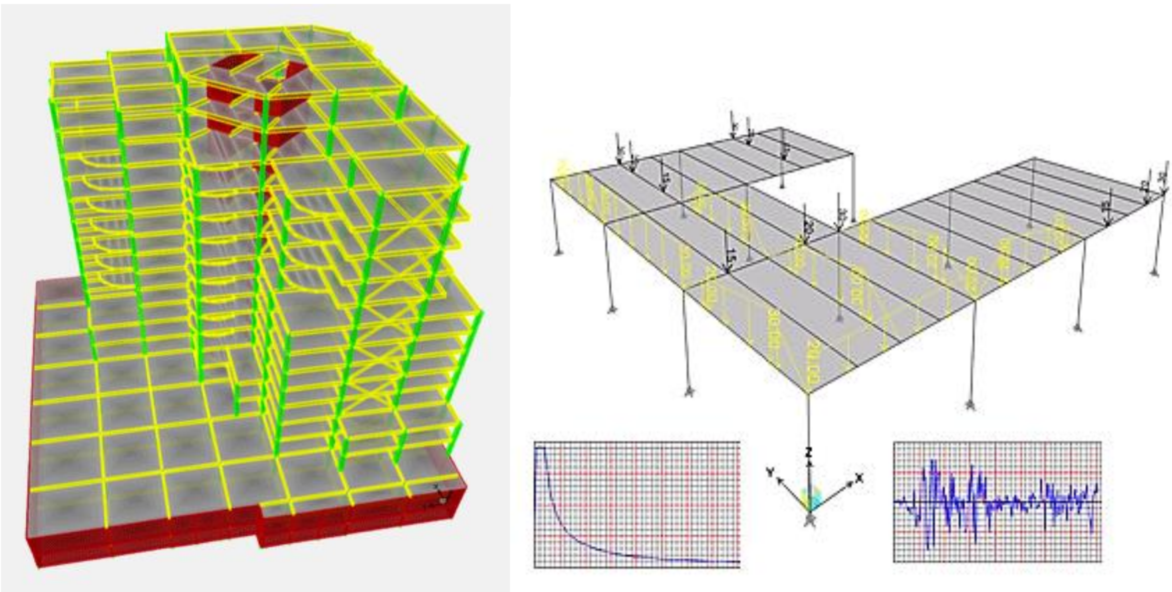


Figure 2.25: view of loading, analysis, and design.

A multitude of structural loads, including as dead, live, earthquake, and wind loads, can be defined by the user of the application. Next, the loads are allocated by the user to the different structural items in the model. It is possible to define an infinite number of load patterns.

For every object with structural qualities, the self-weight of the structure is found by multiplying its volume by its weight per unit of volume.

Once the object has been chosen, you can access the relevant submenu and assignment options by clicking the Assign menu command. The choices and submenus are listed in Table 2.3.

Table 2.3: Load Commands on the Assign Menu

sub menus	**Joint Loads**	**Frame Loads**	**Shell Loads**	**Tendon Loads**
assignment options	Force	Point	Uniform Load Sets	Tendon Loads
	Ground Dis-placement	Distributed	Uniform	Tendon Losses
	Temperature	Temperature	Non-uniform	
		Open Structure Wind Parameters	Temperature	
			Wind Pressure Coefficient	

II.11 Define Load Cases

A load case specifies how the structure is to be loaded and how the structural response is to be computed. Depending on how the model reacts to the loading, analyses are broadly categorized as either linear or nonlinear. Dead and live static loads acting in the direction of gravity make up the loads employed in this tutorial.

To access the Define Loads form displayed in Figure 2.26, click the Define menu > Load Cases command. It should be noted that a dead load case with self-weight (DEAD) is the only default load case that has been established.

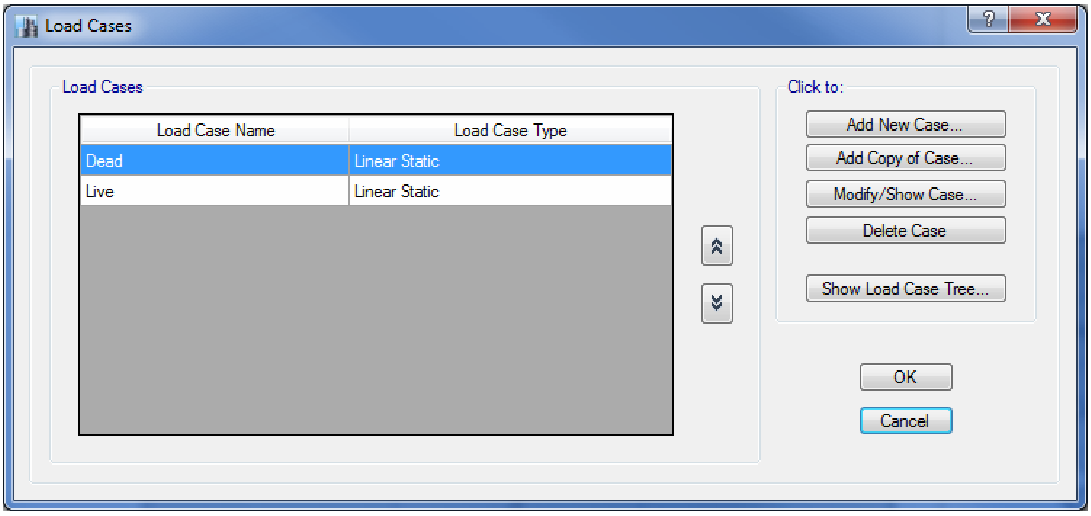


Figure 2.26: Load Cases form

Take note that the default case has the self-weight multiplier set to 1. This means that the load scenario will automatically include each member's self-weight multiplied by 1.0.

Click the Add New Case button, which opens the Load Case Data window seen in Figure 2.27, to establish a new load case.

Click on the Load Name column edit box. Enter the load name, LIVE, for the new load. Using the pull-down option, choose a type of load (in this example, LIVE). Check to see if the Self Weight Multiplier is zero. For the LIVE load to be added to the load list, click the Add New Load button.

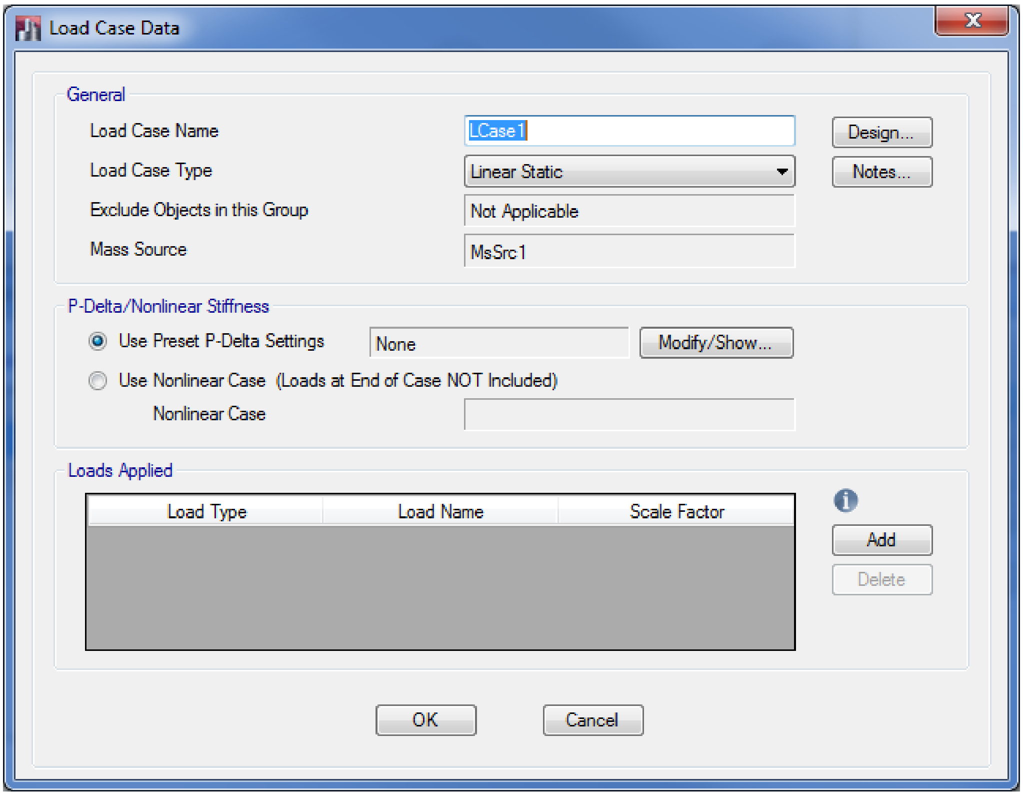


Figure 2.27: Load Case Data form

There are Load Cases and Analysis Cases in Etabs, and they could differ from one another. But when a load case is defined, the program automatically generates a corresponding analysis case, and the analysis cases are accessible for inspection when the analysis is performed.

II.12 Assign Gravity Loads

Here, the model will be subjected to both living and dead gravity loads. Ensure the software is in the choose mode and that the X-Y Plane @ Z=0 view is still active.

A. To choose every object in the deck, draw a selection box from right to left across the whole deck. The phrase "Areas Selected" ought to appear in the status bar in the lower left corner. Click the Clear Selection option to undo your previous selections, then try again.

B. To activate the Uniform (Shell) command, select Area Loads > Assign menu. The Area Uniform Loads form appears as a result. As seen in Figure 2.28, choose DEAD from the Load Case Name drop-down menu.

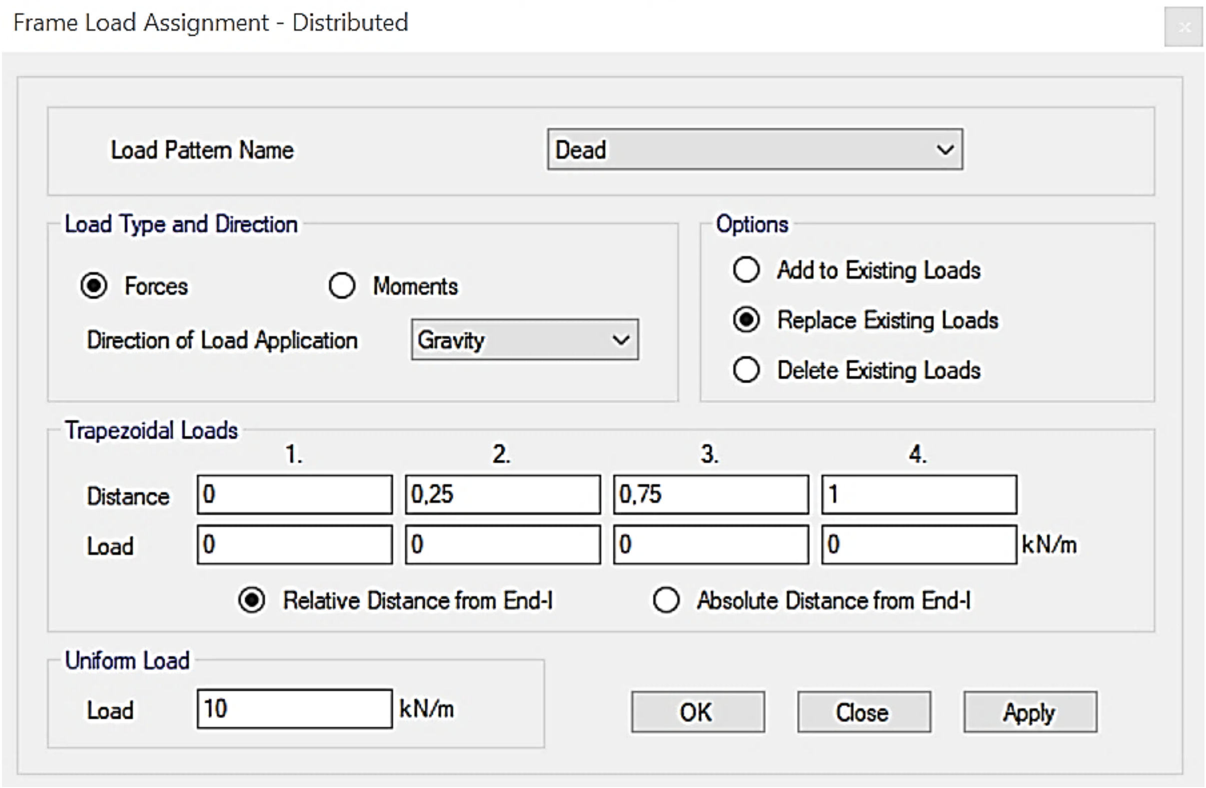


Figure 2.28: Area Uniform Loads form.

1) From the Units drop-down menu, choose KN-m.
2) In the Uniform Load area, type 10 in the Load edit box.
3) Remind yourself that the direction of gravity is in the global Z direction, which is negative.
4) To accept the dead load, click the OK button.

II.13 Analyze of the Model 

In order to initiate the analysis, either click the Run Analysis button or the Analyze menu > Run Analysis command, or click the Run Now button if the Set Load Cases to Run form is still visible.

When the Analysis Monitor is set to "Always Show" or "Show After," the software will display a window that says "Analyzing, Please Wait." The application will execute the analysis as data scrolls in this window.

II.14 Discretization

The process of converting an object-based model's material domain into an analytical model fit for analysis is known as discretization. Discretization in structural analysis can use one of two primary categories of analytical models, such as [6]:

Node-element model: wherein distinct lines joined by nodes serve as the representation of the structural elements. Each node has six degrees of freedom in a three-dimensional system, which can be limited or free. Line elements mimic the physical behavior of structural elements by adhering to mathematical relationships, hence characterizing their geometric and material qualities. The direct stiffness method is used to translate loading at node positions into displacement and stress fields, which show the structural performance.

Finite-element model: wherein a material continuum is joined by nodes to form a network of line elements through a meshing process. The geometric and physical characteristics of the local material are simulated by each line element. The computational model may allow for the numerical formulation of structural response given the loading and boundary conditions of the entire system. A finite-element model's discretization will result in a coarse or fine mesh depending on how refined it is. Technically speaking, a node-element model is a finite-element model where the structural element is represented by a single line element. However, while finite-element modeling employs the finite-element method (FEM), node-element modeling adheres to the direct stiffness technique.

Division of frame elements: Although discretization of an object-based model is usually necessary (as it makes analysis easier), there are some situations where it's also crucial to split frame elements into many segments in order to get accurate findings. [6].

Divided frame elements are helpful for the following kinds of analyses:

- Buckling analysis: To catch higher modes.
- Dynamic analysis: Because mass is assembled at joint points, this method better captures mass dispersion.
- P Delta (P-δ) effect: To analyze equilibrium circumstances regarding displacement configuration, better capture local column deformation.
- Displacement accuracy: Values are interpolated from the nodes at either end of the frame element to form joints when precise displacements are required.

II.15 Meshing area

A mesh is a set of connected line elements and nodes that is used to simulate the behavior of a structural system under applied loading and solve numerically. In order to simulate the material and geometric properties of the structural system, computational techniques first generate an analytical model by populating the material domain with a finite-element mesh in which each line element is assigned mathematical attributes (axial, bending, shear, and torsional stiffness, for example). After that, the system is forced to operate within boundary constraints and is exposed to either mechanical or thermal loads. After then, structural stresses, strains, and displacements may be resolved numerically [7].

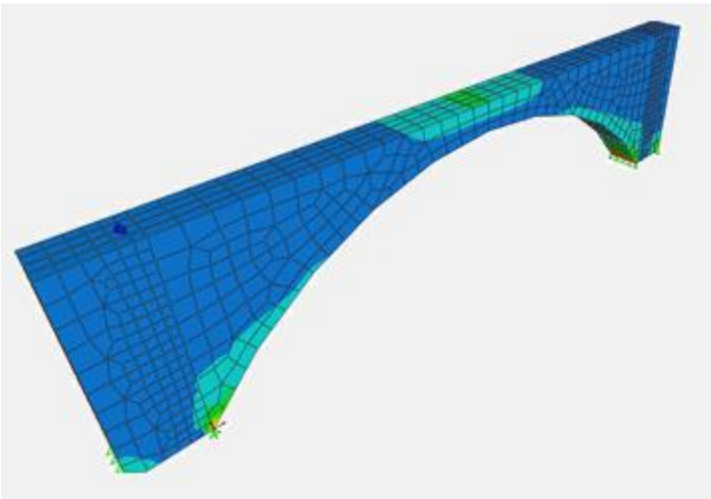


Figure 2.29: example of area mech

II.16 Damping

Damping affects dynamic reaction and is a quality of both the material and structure. For every kind of load scenario, a certain kind of damping is offered. A fixed amount of damping is applied uniformly to every load case of that kind, while certain load instances may require more damping [7].

- For response-spectrum and modal time-history investigations, modal damping is employed. Weighting is based on element and modal stiffness in material modal damping, also referred to as composite modal damping. User-specified material modal damping ratios (r, where $0 < r < 1$) correspond to the damping ratio of each mode, are given for each material.
- **Viscous proportional damping**: Direct-integration time-history analysis is performed using viscous proportional damping. This characteristic varies with stiffness and mass.
- **Hysteretic proportional damping**: For assessments of power spectral density and steady state conditions, hysteretic proportional damping, along with mass and stiffness, is employed.
- **Damping devices** : Another way to model damping devices is as a structural subsystem.

II.17 Rigid vs. Semi-rigid diaphragm

Joints inside a plane are linked by a diaphragm constraint to move as a planar diaphragm that is resistant against membrane (in-plane) deformation but vulnerable to plate (out-of-plane) displacement and related consequences. When floor diaphragms are modeled with extremely high in-plane stiffness, numerical accuracy issues are alleviated by diaphragm limitations. By decreasing the size of the eigenvalue formulation, they also improve the computing efficiency of dynamic lateral analysis. [8].

While semi-rigid diaphragms imitate real in-plane stiffness properties and behavior [9], rigid diaphragms exhibit membrane deformation and report the corresponding forces because of their infinite in-plane stiffness qualities [10]. Rigid diaphragms yield results that are almost equivalent to semi-rigid diaphragms' while benefiting from faster computation for the majority of reinforced-concrete slab systems when the slab is thick enough and membrane deformation from lateral stress is minimal. When substantial in-plane deformation does occur, or when mandated by code, semi-rigid diaphragms ought to be represented. Only components in the same X-Y (horizontal) plane are eligible to use floor diaphragms.

It is possible to construct semi-rigid or flexible diaphragms in Etabs by explicitly modeling the floor slab using area objects.

CHAPTER 3:

STUDY AND MONITORING OF A REAL PROJECT USING ETABS 20.3

CHAPTER 3: STUDY AND MONITORING OF A REAL PROJECT USING ETABS 20.3

III.1 Introduction

Modeling on Etabs 20.3 consists of the following steps:

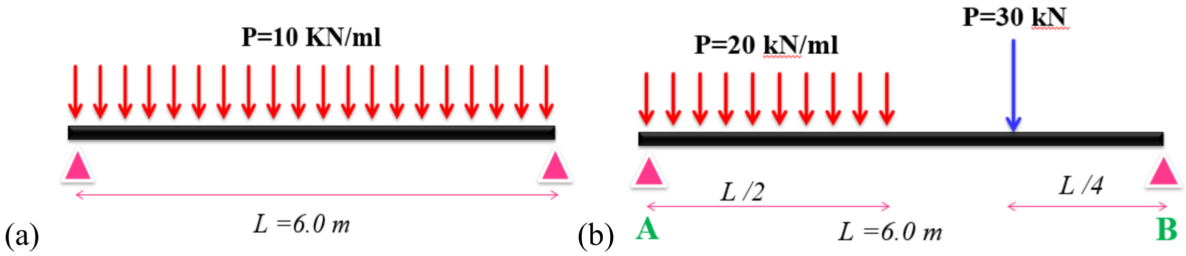
1) Enter the model's shape, including the nodes' locations and the elements' connections.
2) Describe the attributes of the constituents and allocate them to the constituents.
3) Assign the loads to the nodes and elements by defining the load situations (static and dynamic).
4) Define the boundary conditions (diaphragms, supports, etc.).
5) Begin solving the problem and, if needed, make adjustments to the model.
6) Examine the results (on a file, the screen, etc.).

III.2 Examples of applications over 1D continuous beams

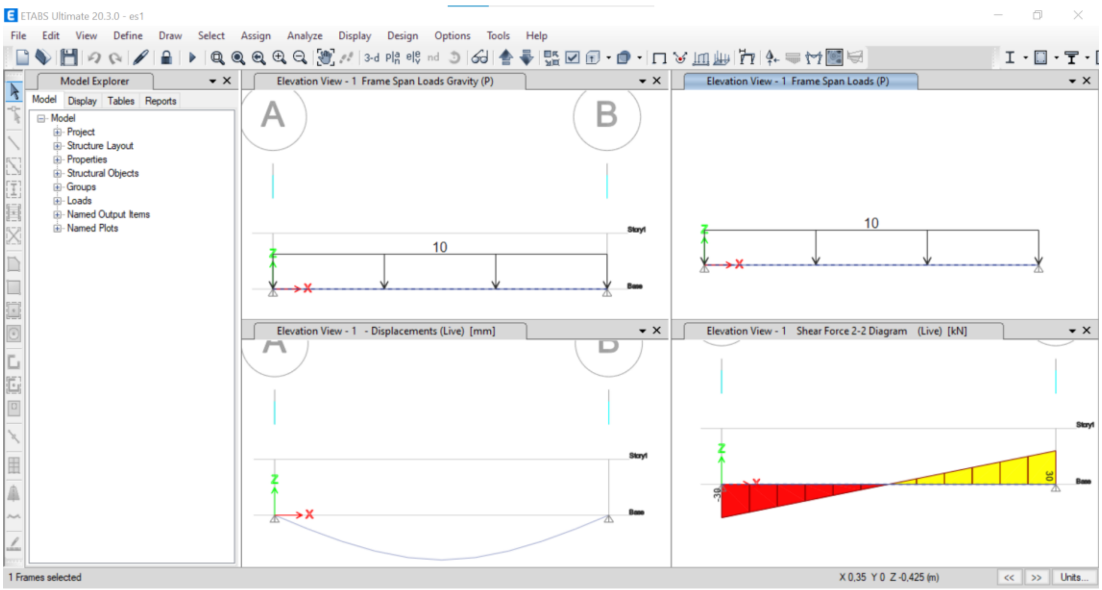
The finite element formulation for axially loaded bars, planar elasticity issues, axisymmetric solids, and general three-dimensional solids is the foundation used by Etabs software for structural analysis. The mathematical solution is described in the book of "Structural Analysis with the Finite Element Method. Linear Statics, volume 2" [11].

We ask in the following examples to draw the diagrams of the shear force and the bending moment and to identify the maximum values of M, T, and V with the use of the Etabs software. In the models that follow, it is considered that the self-weight of the beams is always negligible.

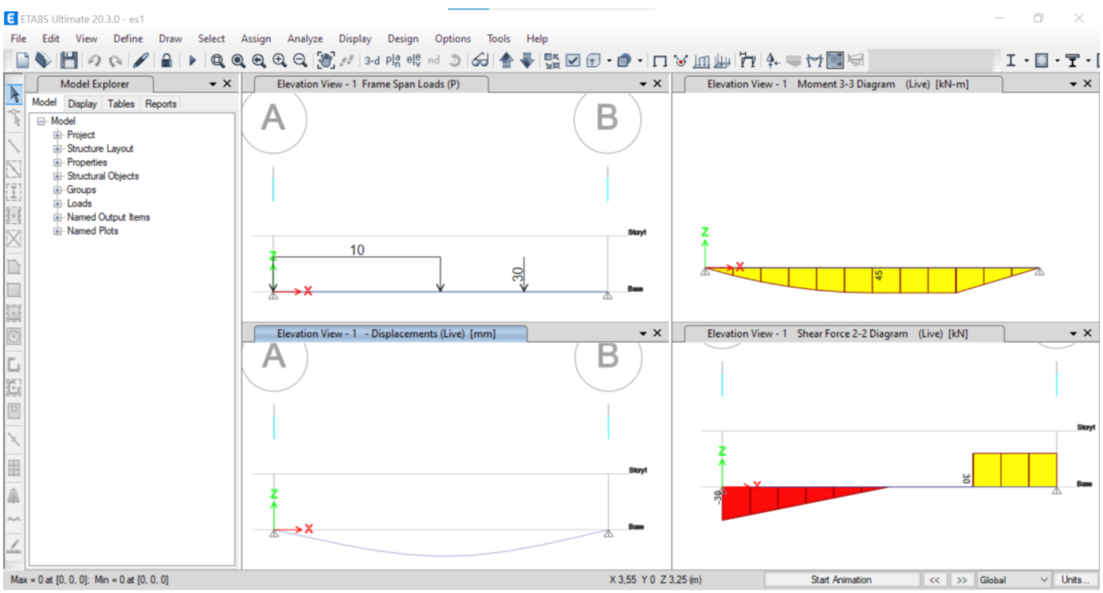
Example 1 : Beams two supports



here is the solution of example 1 under Etabs version 20.3.0

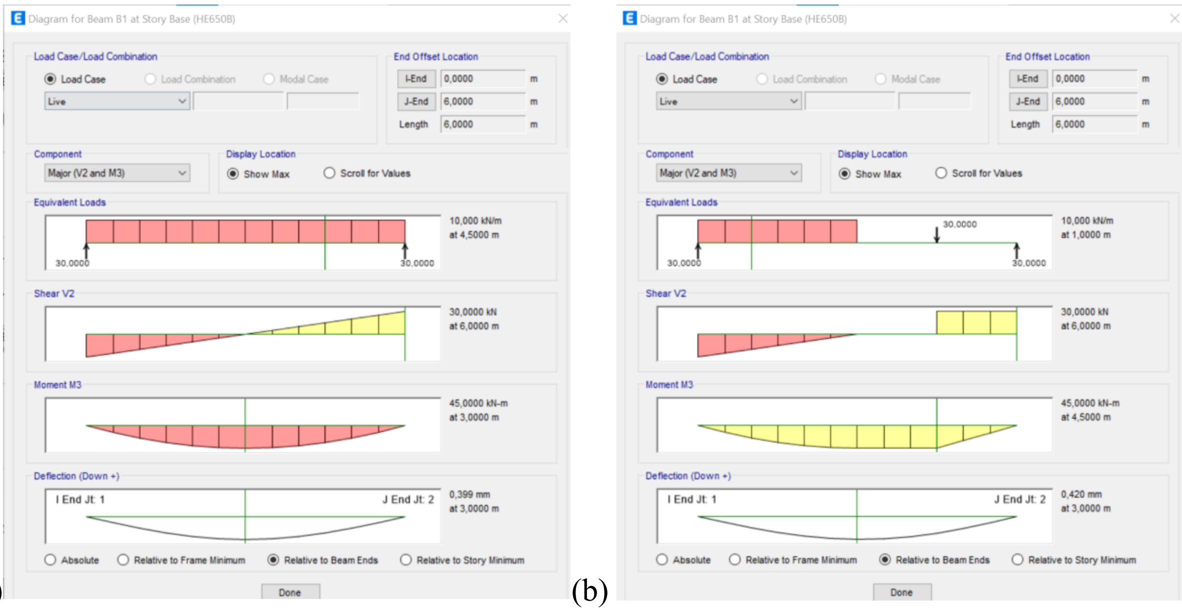


(a)



(b)

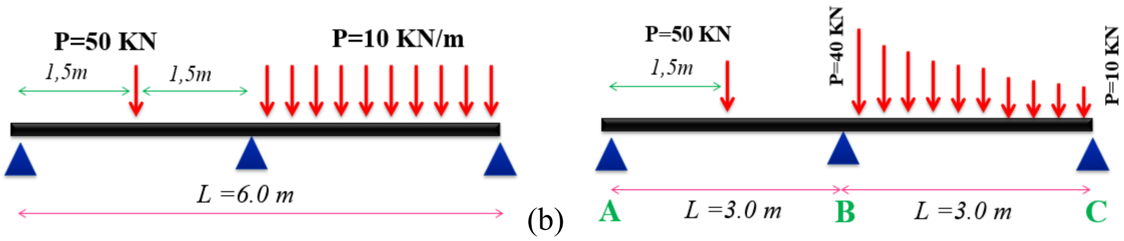
Figure 3.1 : Etabs 20.3 home interface



(a) (b)

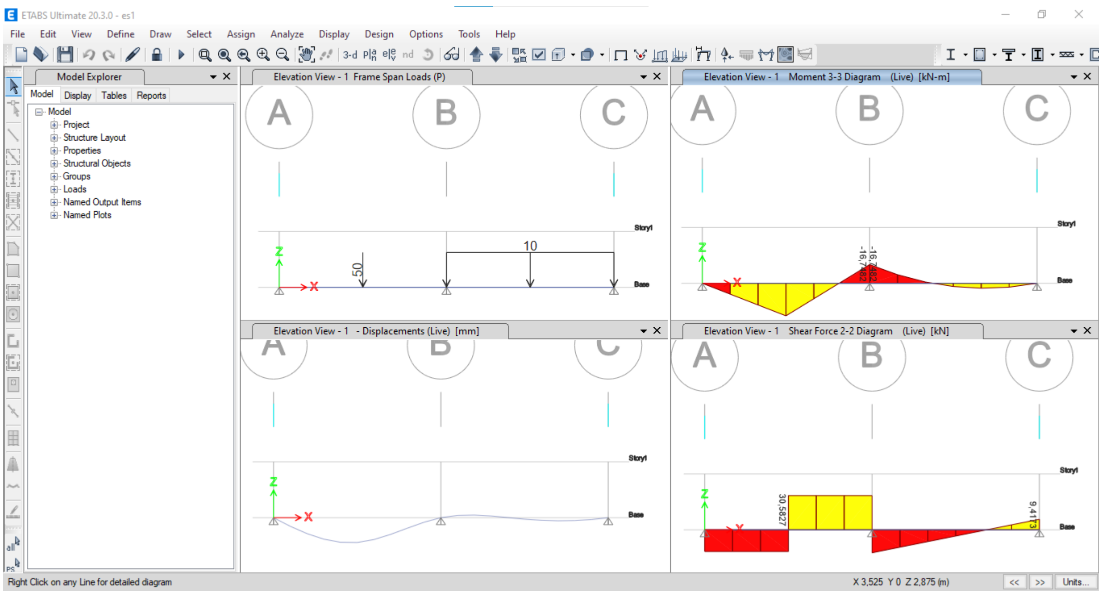
Figure 3.2 : Etabs 20.3 home interface

Example 2 : Beams three supports



(a) (b)

This is how to solve Example 2's two-frame problem using Etabs version 20.3.0.



(a)

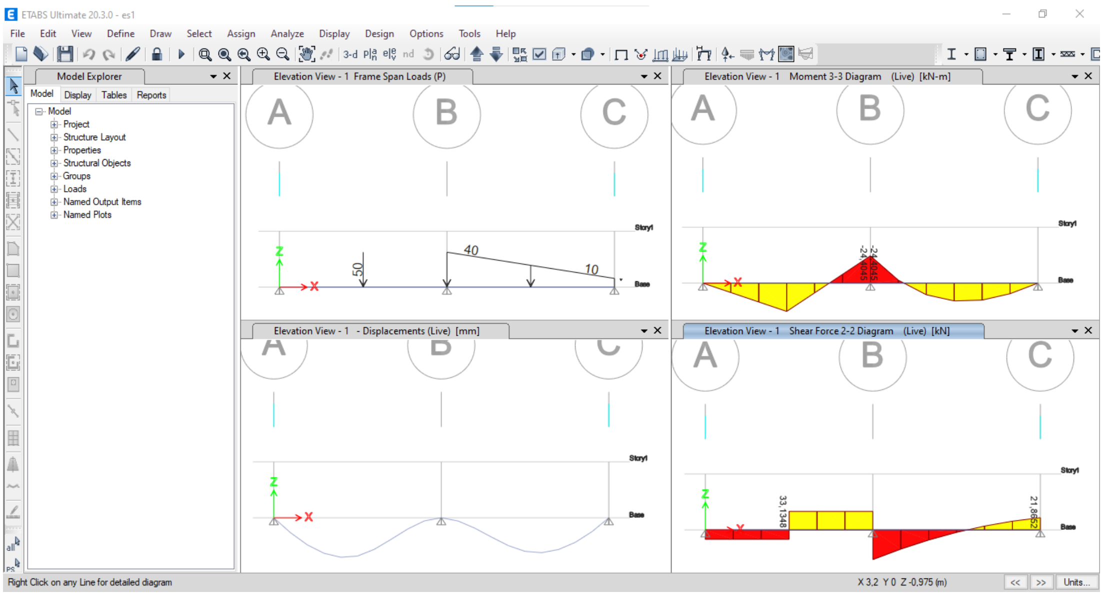
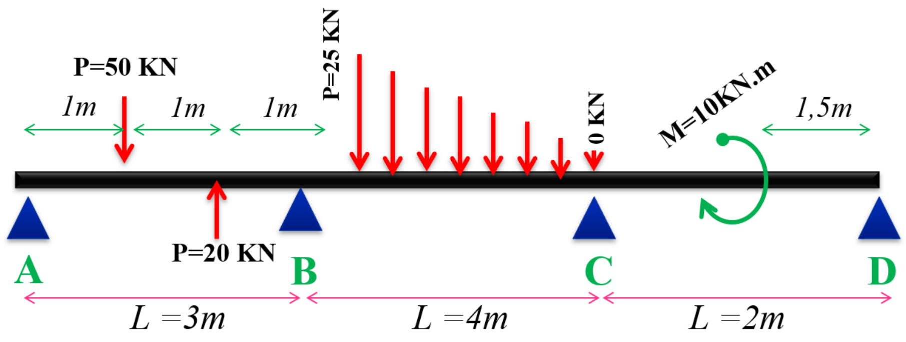
(b) 

Figure 3.3 : solution of the Beams with three supports

Example 3 : Beams four supports



The following figure illustrate the solution of Example 3 of the beam with 4 supports under Etabs version 20.3.0

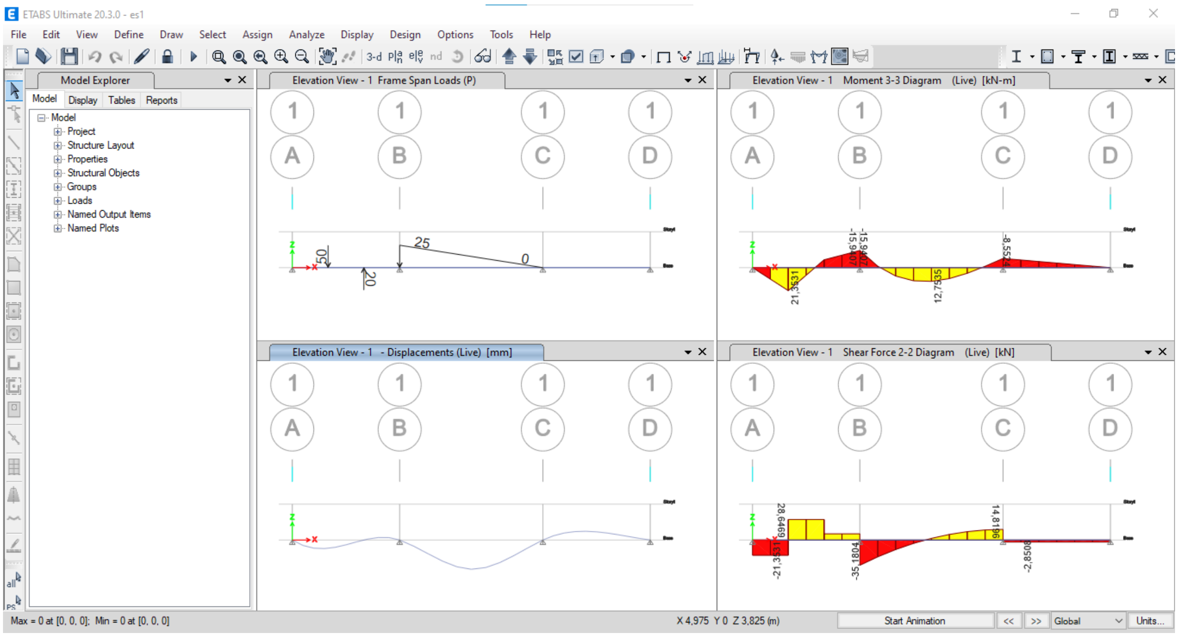
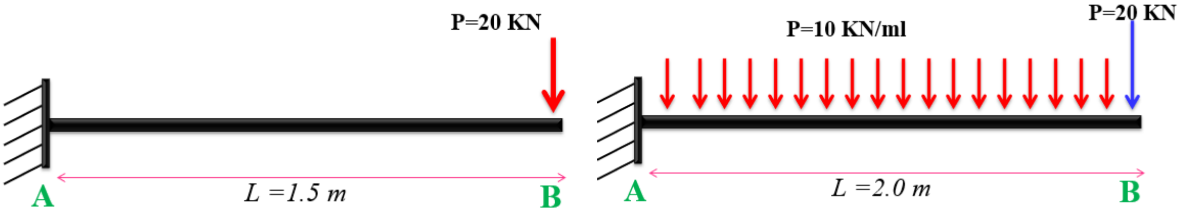
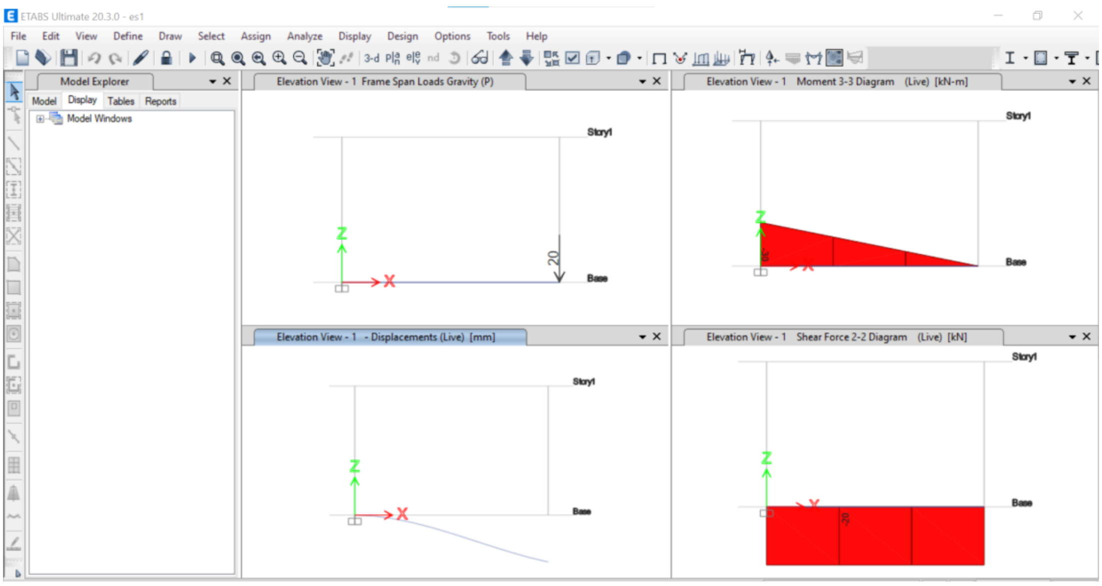


Figure 3.4 : solution of the Beams example with four supports

Example 4 : Beam with Bracket





(a)

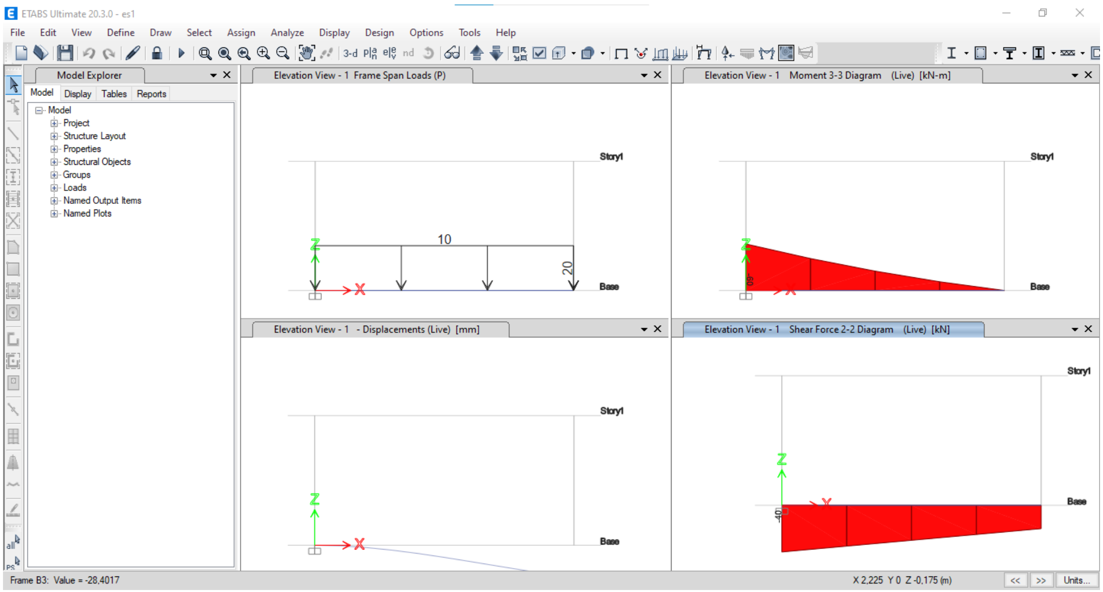
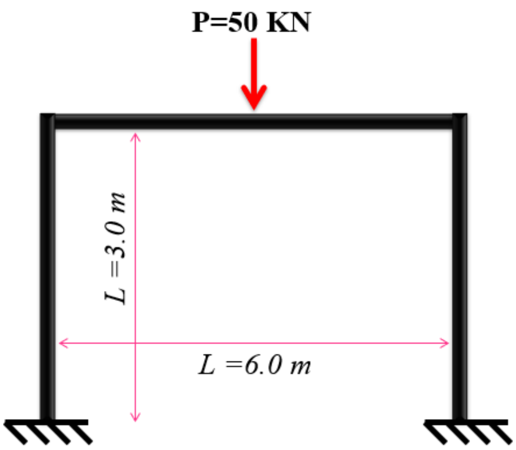
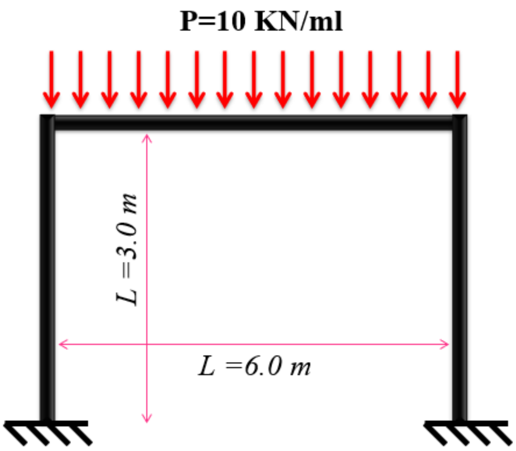
(b) 

Figure 3.5 : solution of the Beams example with Bracket

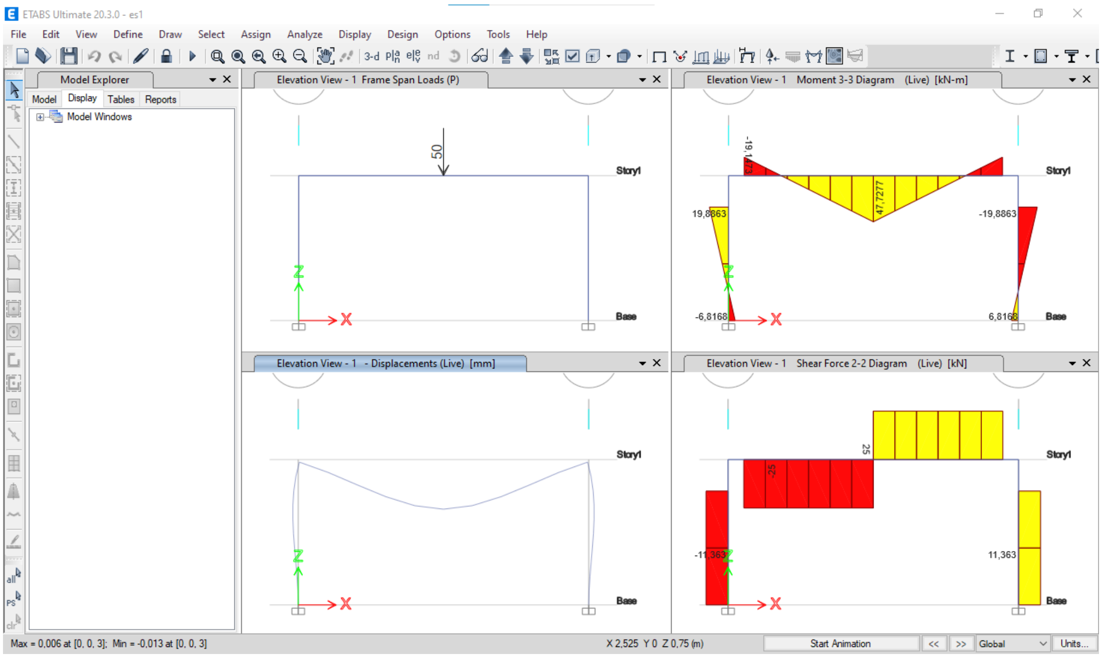
III.3 Examples of two-dimensional frames

Example 5 : Frame 2D with two fixed supports

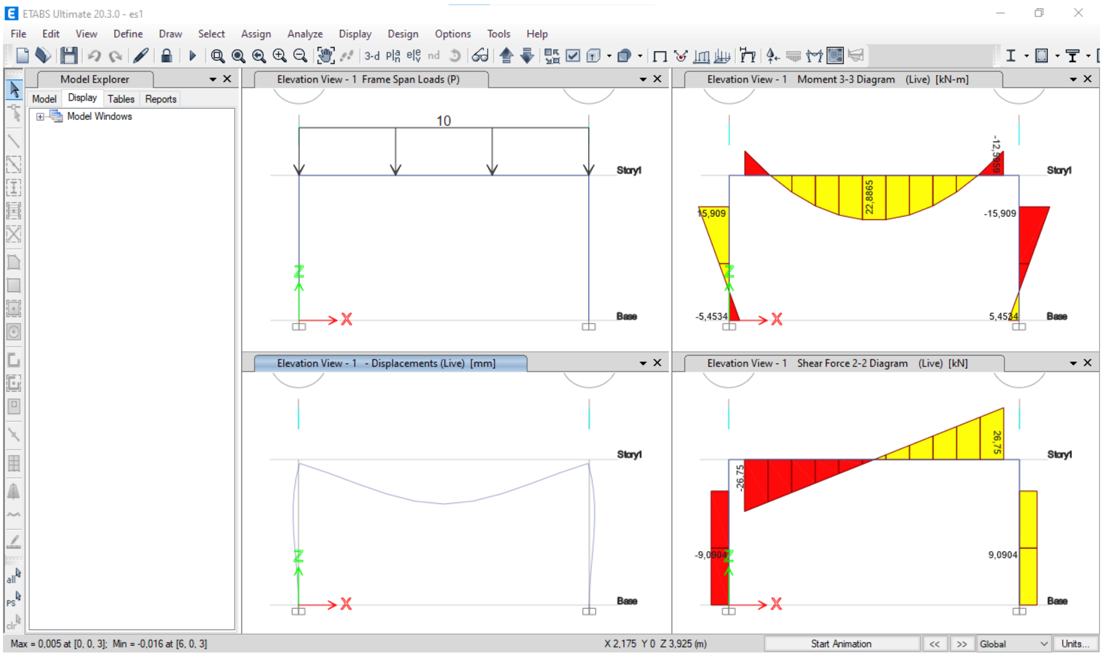
(a) 

(b) 

The solution for example 5 of the 2D frame with two fixed supports and one floor under Etabs version 20.3.0 is shown in the accompanying figures.



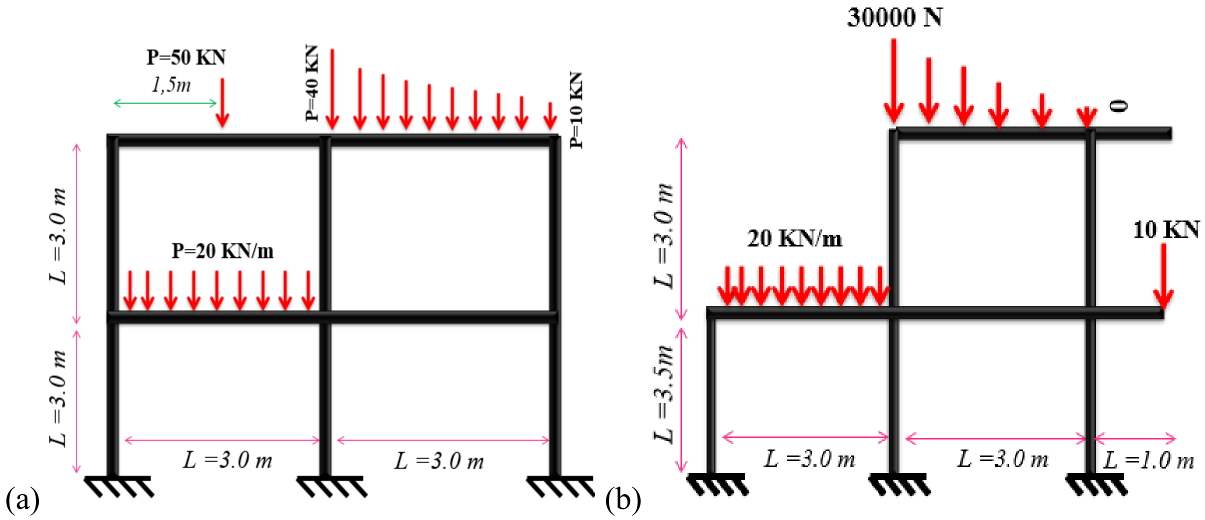
(a)



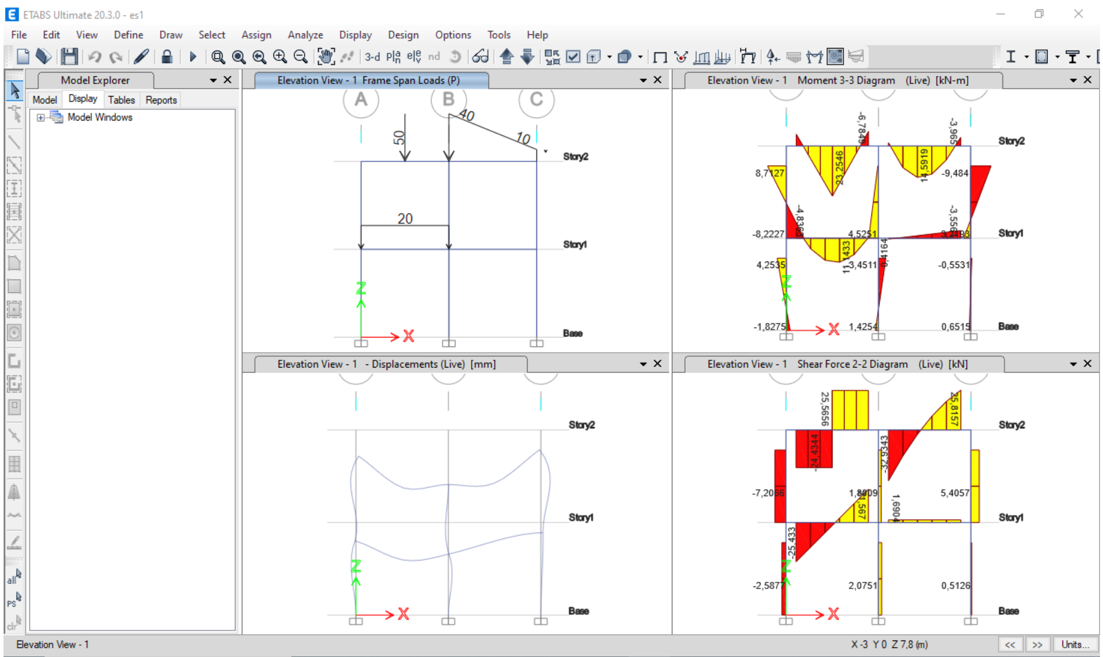
(b)

Figure 3.6 : solution of the exercise of Frame 2D with two fixed supports

Example 6 : Frame 2D with two floors



The following figures show the solution of Example 6 of the 2D gantry with 3 recesses and two floors under Etabs version 20.3.0:



(a)

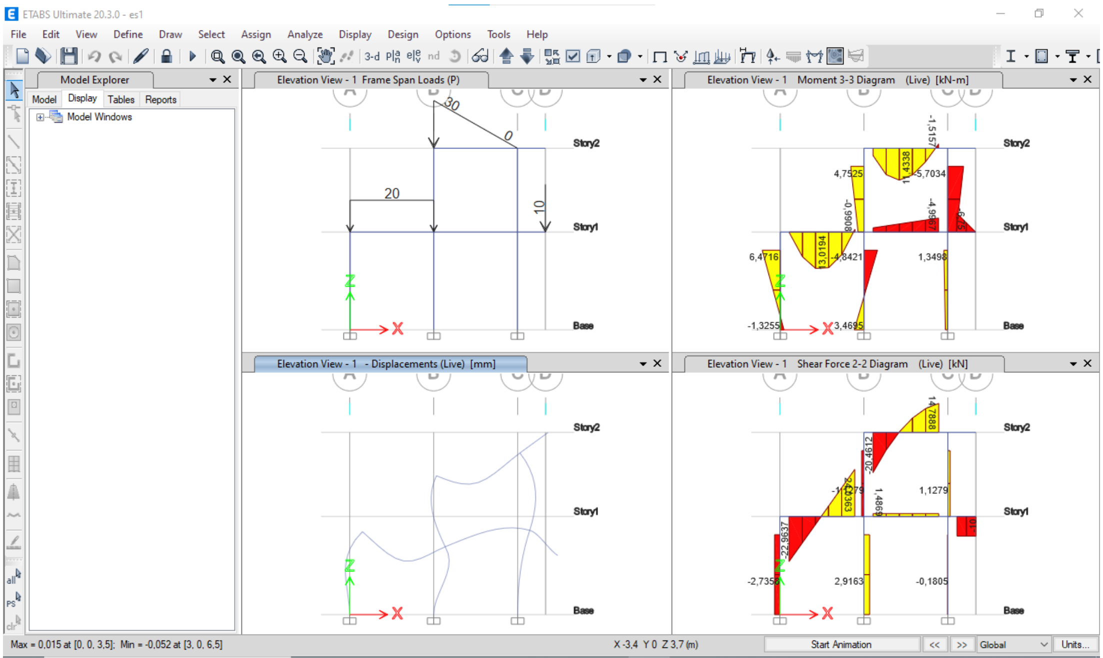
(b) 

Figure 3.7 : solution of the exercise of Frame 2D with two floors

Note: It is noted through these variation examples that the different versions of the Etabs software can solve any type of RDM exercises. Nevertheless, the user must know the theoretical basis within the programming black box.

III.4 Self-stable frame with reinforced concrete for R+4 floors

The structure studied is a reinforced concrete structure for housing usage situated in region 2A according to the Algerian seismic regulation RPA99 version 2003 [12]. The dimensions of the structure are given in the following figures.

Length (XX) .... 18.30 m

Width (YY) .... 08.80 m

Height (ZZ).. 16.00 m

Floor heating ………………………………………………………………………. 03.20 m

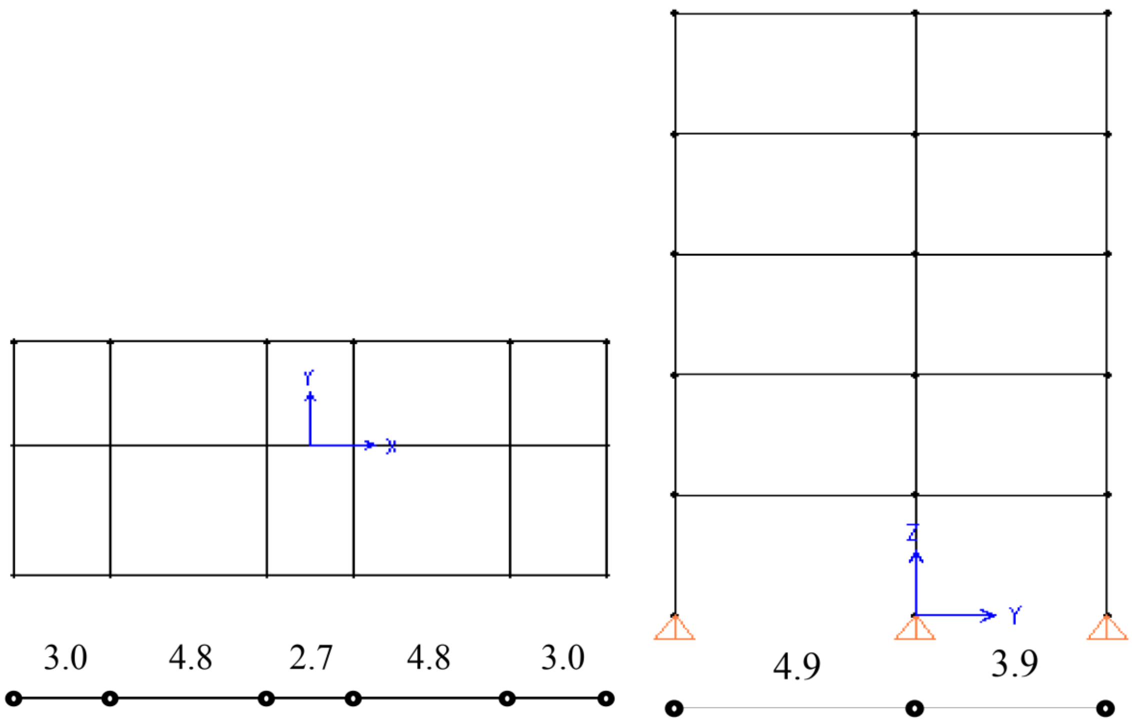


Figure 3.8: Plan and elevation dimensions of the studied structure.

The load relating to the self-weights and to the permanent load are denoted by G. The live loads are denoted by Q and generally represent the overloads.

For the permanent charges G we have:

- Terrace Floor G = 5.50 KN/m^2
- The current Floor G = 5.00 KN/m^2

And for the live load Q we have:

- Terrace Floor Q = 1.00 KN/m^2
- The current Floor Q = 1.50 KN/m2

For load combinations, we have two case; first the ultimate limit and second the service limit (ELU and ELS) [13]:

- ELU = 1.35 Dead + 1.5 Live
- ELS = Dead + Live

Step N° 01: Define the Geometry of the Model

The first step consists in defining the geometry of the structure to be modeled.

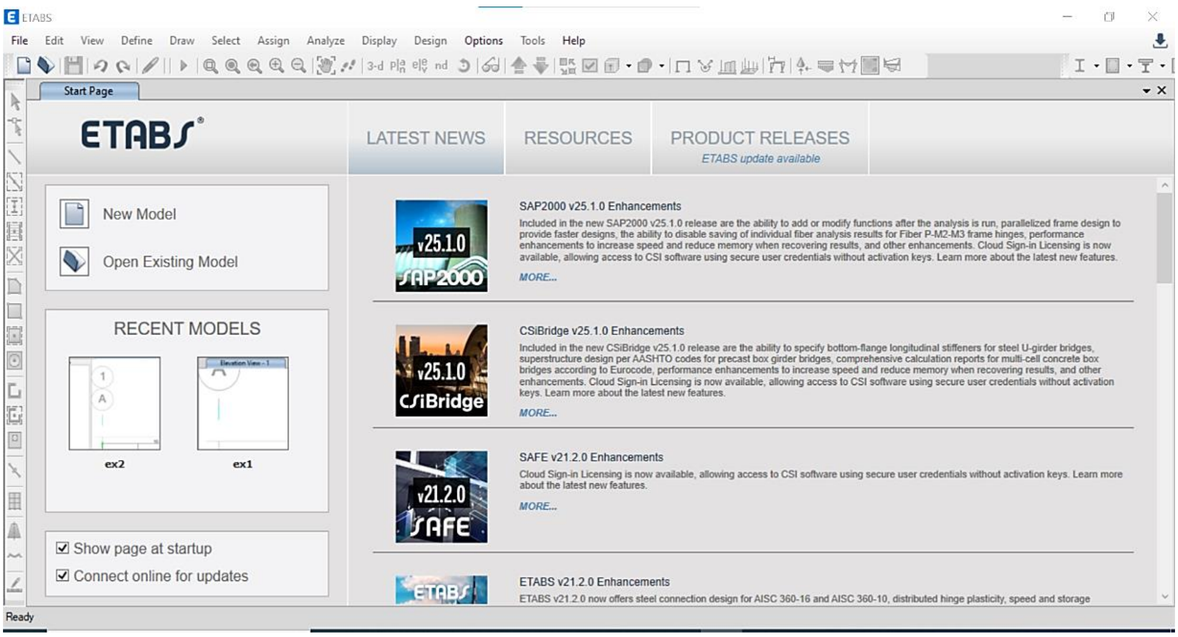


Figure 3.9 : Etabs 20.3 home interface

Choice of Units: You must choose a system of units for data entry in Etabs. In the screen that appears, select, for example, the unit (MKS) as the basic units for forces and displacements. And given the absence of the Algerian construction code, we will choose Eurocode 3 and 2 for the calculation of reinforcement [14].

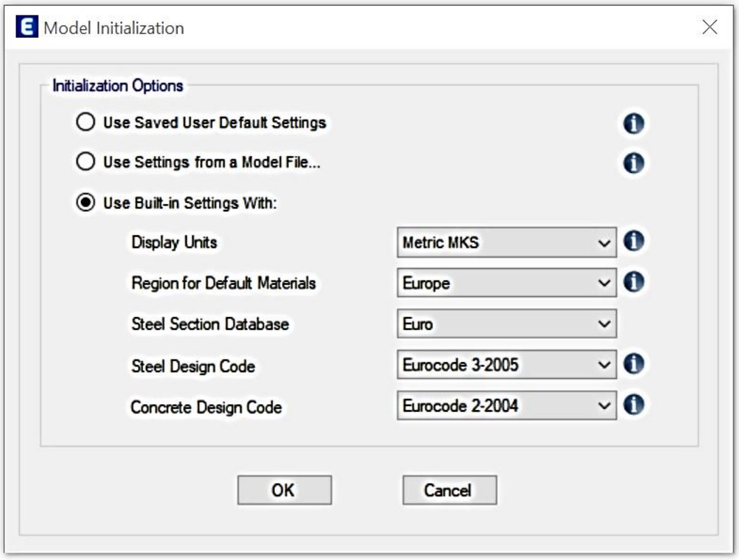


Figure 3.10: Choice of system of units for solicitation and displacements

Basic Geometry:

Select ***New model*** from template from the ***File*** menu. This option allows you to quickly create a "regular" template, using predefined templates.

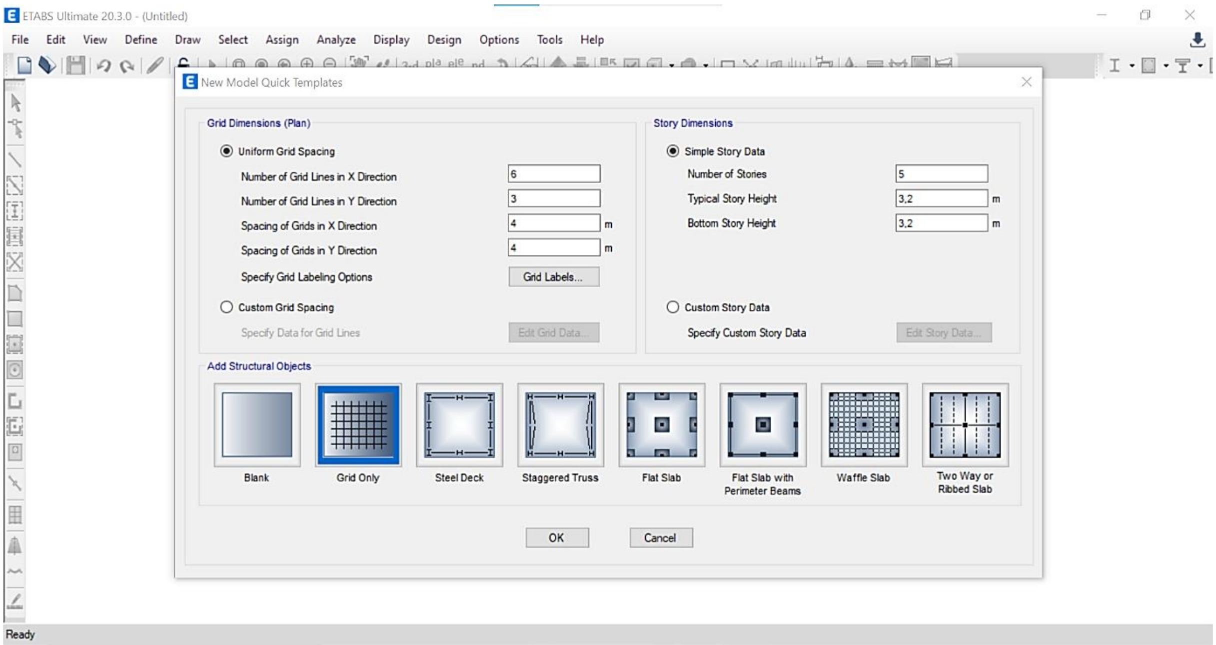


Figure 3.11: menu to introduce the basic geometry

You must specify the number of floors, the number of spans following X and Y, height of the floor and the width of the spans following X and Y. Enter the previous data, even if the building is irregular (changes will be made later). The following window presents the data from our example.

Save your model using the File and Save menu. You must specify the backup folder and the file name. The SAP2000 data files have "SDB or S2K" extensions. You can also click on the icon .

Changing the Basic Geometry

As the geometry of the building is not quite regular, it must be modified. The length of the spans is different. Select ***Draw*** and ***Edit Grid***, on the window that appears on the screen, choose the X axis and replace the coordinate line X=-12 by X=-9.15 by clicking on ***Move Grid Line***. Make sure that the Glue Joints to ***Grid*** Lines option is selected.

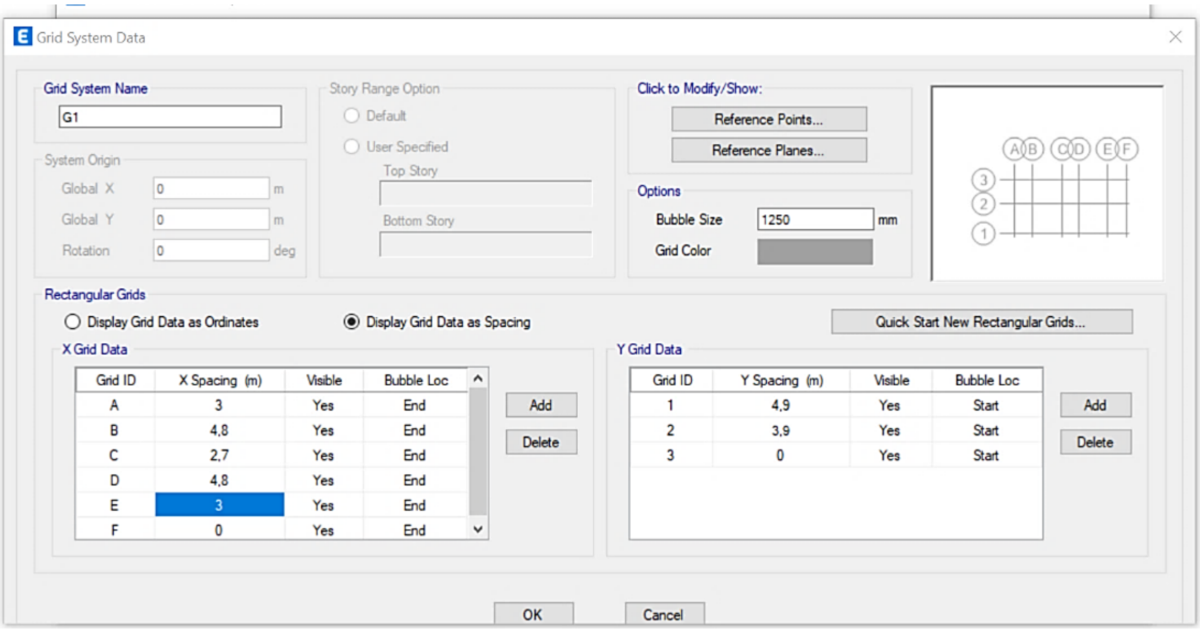


Figure 3.12: menu for modifying the grid lines for structures with different frames.

Repeat this operation to replace the coordinate lines 4.0; 4.0; 4.0; 4.0; and 4.0 by 3.0; 4.8; 2.7; 4.8 and 3.0 respectively.

Select OK and repeat this operation for the Y and Z directions as follows: Y direction replace the line Y=4.9 by 3.9, Select OK. For the height of the floors the dimensions are as follows; Z=3.2; 6.4; 9.6; 12.8 and 16, Select OK.

Your template should now like this :

Two views of the model are displayed (3D and 2D). If you open a window and the model is no longer centered, click on the Restore full view icon.

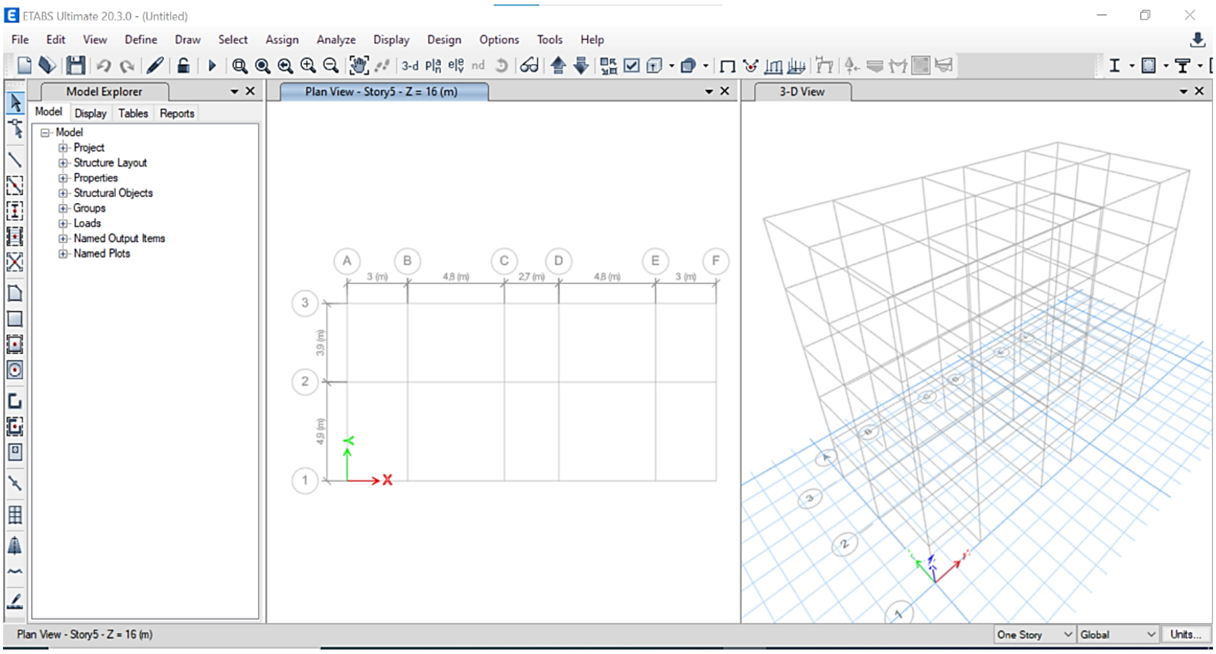


Figure 3.13: display of the coordinate system for the modeling of the structure

Step N° 02 : Specification of the Properties of the Elements

The second step is to specify the properties of the different elements of the structure such as posts, beams, slabs and sails if they exist.

Definition of the material

Select ***Define*** and ***Materials*** then ***Modify/Show Material*** to view or modify the characteristics of a material already existing in the Etabs 20.3 library or ***Define and Materials*** then ***Add New Material*** to add a new material, in both cases,

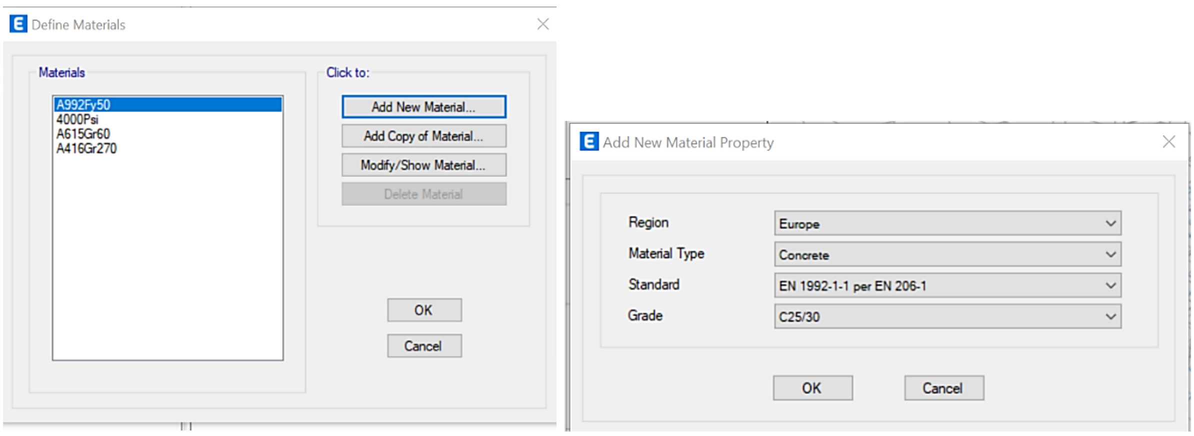


Figure 3.14: Menu for defining materials in Etabs 20.3

In both cases, when you click ***Ok***, this window appears where you must specify for the material the name in ***Material Type***, the type in Type of Material, the density, the density weight, the modulus of elasticity, the poisson's ratio and the coefficient of thermal expansion in ***Material Property Data***.

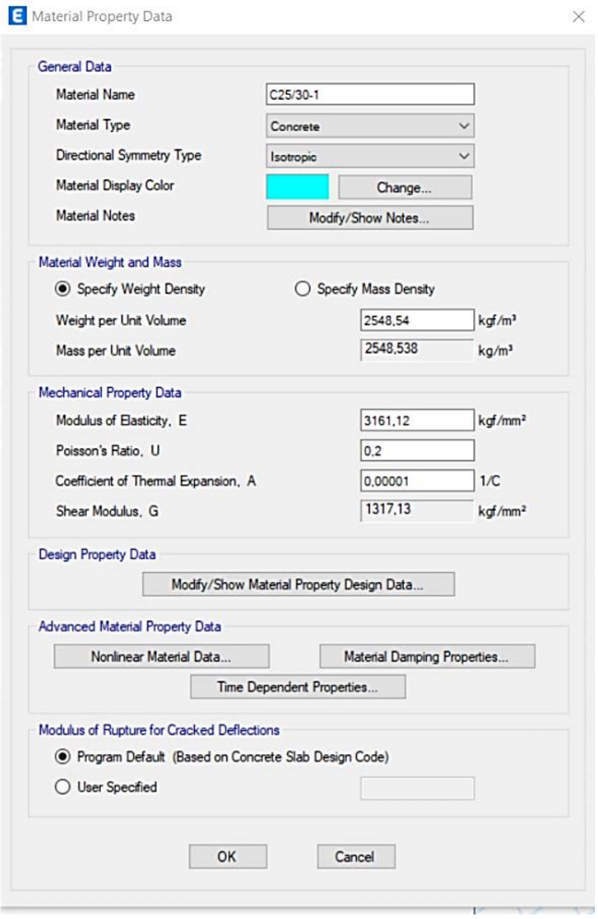


Figure 3.15: Menu to modify the material properties in Etabs 20.3

Choice of Sections

There are a multitude of predefined sections in Etabs 20.3. It is possible, for example, to choose from a long list of steel profiles containing all the information for a given section (IPE, HEA, UPN, W, etc ...). For the example considered here, as the sections are not standard, it is first necessary to define the new sections for the beams and coulombs. We must then assign them to the corresponding elements.

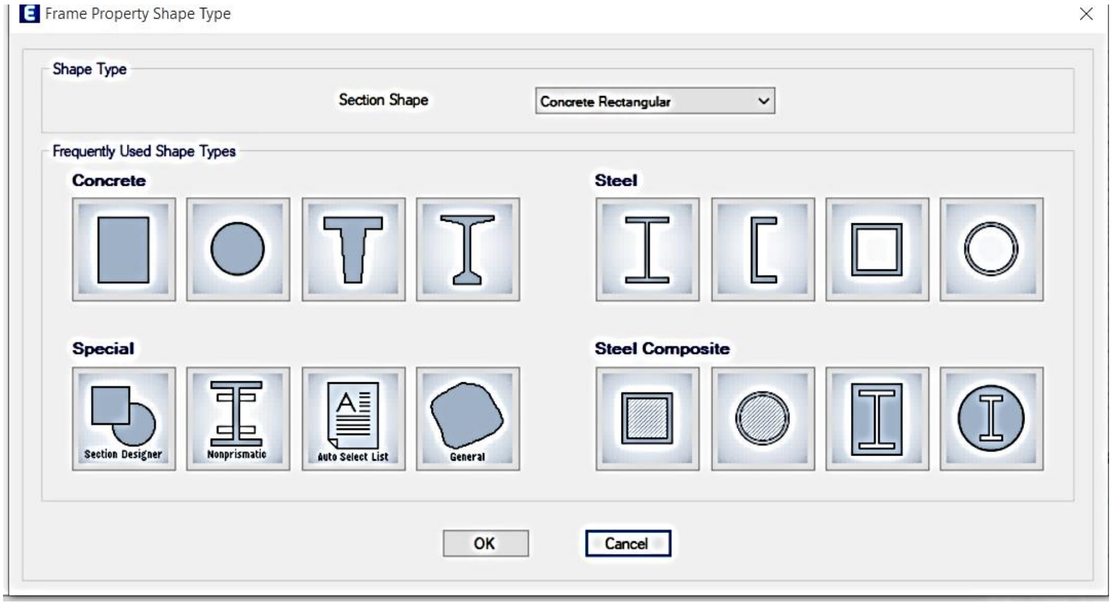


Figure 3.16: Choice of concrete or steel sections in Etabs 20.3

Select ***Define*** and ***Frames Sections*** then ***Import/Wide Flange*** to import a section predefined in the Etabs 20. or ***Define*** and ***Frames Sections*** then ***Add Rectangular*** to define a rectangular section.

The following window appears, you must specify for each section the name in ***Section Name***, the type of the material in Material and the dimensions (width and height) in Dimensions. So, choose Poteau 35 as the section name of the ground floor and 1st floor coulomb of dimensions 35x35 cm and enter the dimensions, in millimeter, on both sides (350x350mm). Select the Beton 25 for the concrete material.

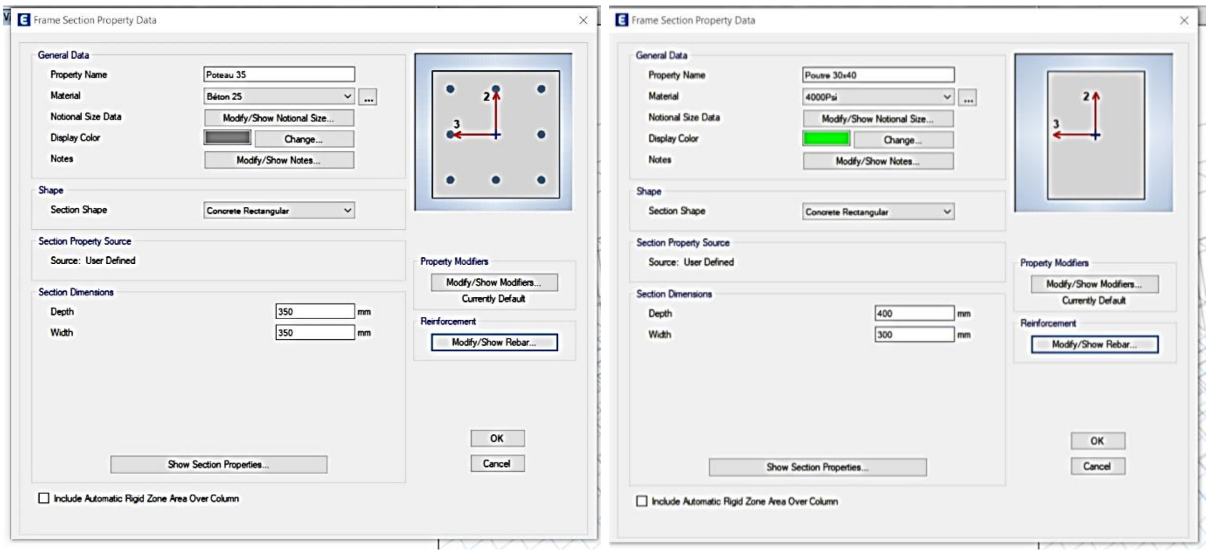


Figure 3.17: definition of the geometry of the coulomb on the left and the beams on the right.

Note: Click on the ***Properties Section*** button to view the area, moments of inertia, shear area and other properties calculated by Etabs 20.3.

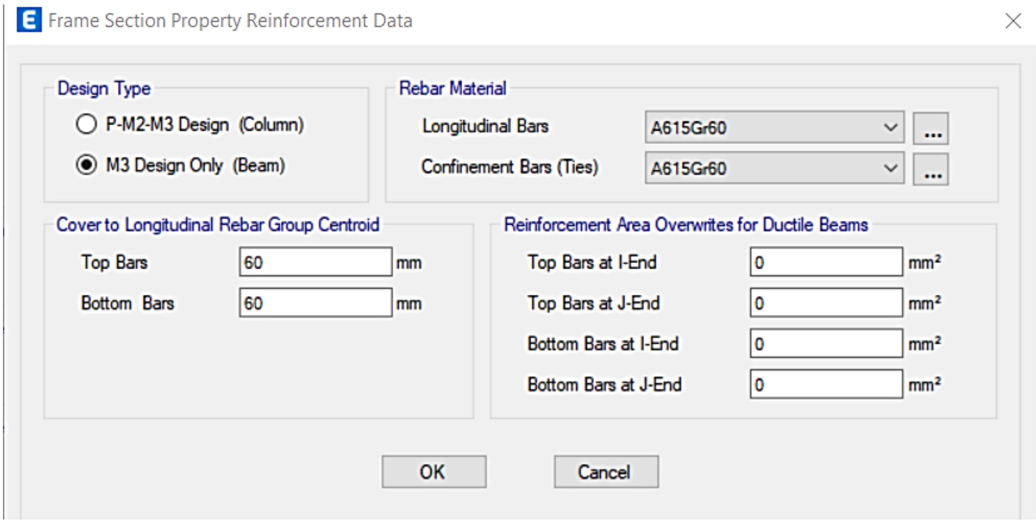


Figure 3.18: Frame Section Property Reinforcement Data menu.

Click "OK" and repeat this operation by choosing as the section name: Poteau 30 to define the section of the coulomb with dimensions 300x300 mm

Step N° 03: Assignment of Sections to Structural Elements

The sections Poteau 35, Poteau 30, Beam 40X30 have been defined and the Concrete material 25 corresponds to the properties of the concrete of the building considered. Now we must assign these properties to the appropriate elements.

To assign the Poteau section 30 to the ground floor and 1st floor coulomb, for example, present the building structure in the XZ or XY plane on one of the displayed model views (3D and 2D) on screen. Click on the ***Clear*** icon in the floating toolbar to eliminate any selection.

Note: This icon is only active if there is already a selection made.

Select in the chosen plan the posts concerned (1st and 2nd floor posts) using the icon in the floating toolbar . This allows you to draw a line with the mouse and select all the elements crossed by the line. Repeat this operation for the poles belonging to the planes parallel to the chosen plane using the icons . We assign the sections with the ***Assign*** menu, then ***Frame and Sections***. In this case, choose the POT35 section that suits the selected poles. By clicking on "OK" the name of the section will appear on the active window. Repeat the operation for all the posts of the structure by choosing the appropriate sections.

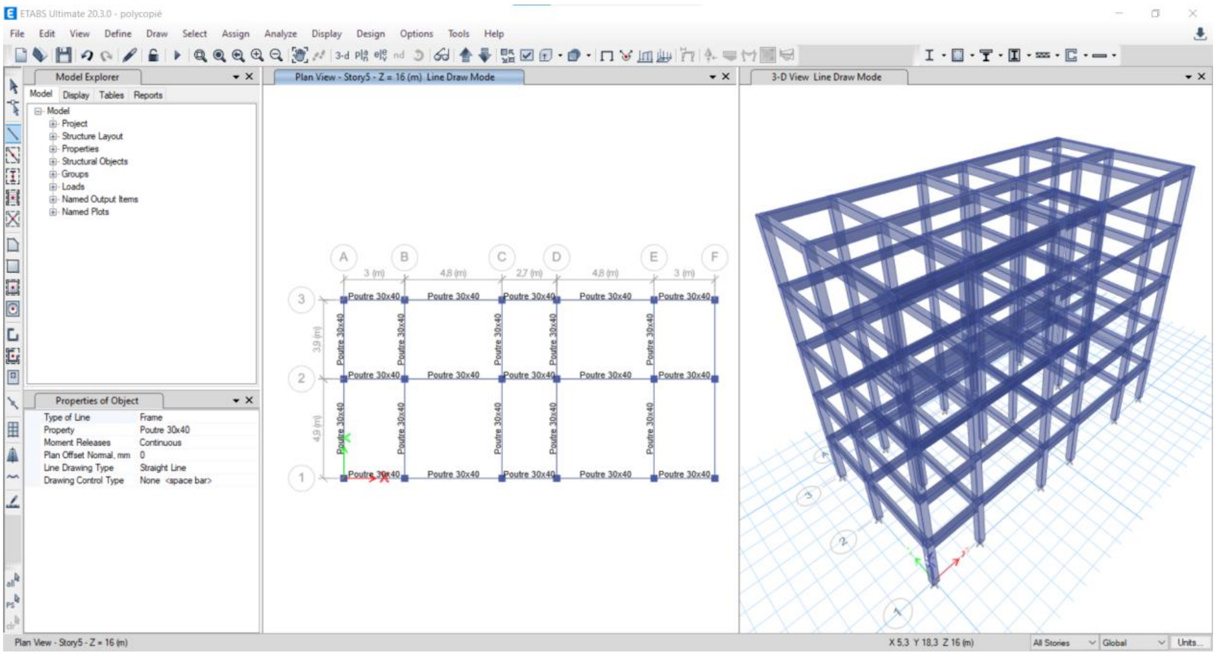


Figure 3.19: Assignment of sections to structural elements in Etabs 20.

This operation is also used to assign the sections already defined to the beams, however, to select the beams, we must, this time, represent the structure of the building in the XY plane.

Model Information Display

It is possible to display different information (node numbers, elements section, bar name, ... etc.) on the model. Click on the Set elements icon  choose the following display options :

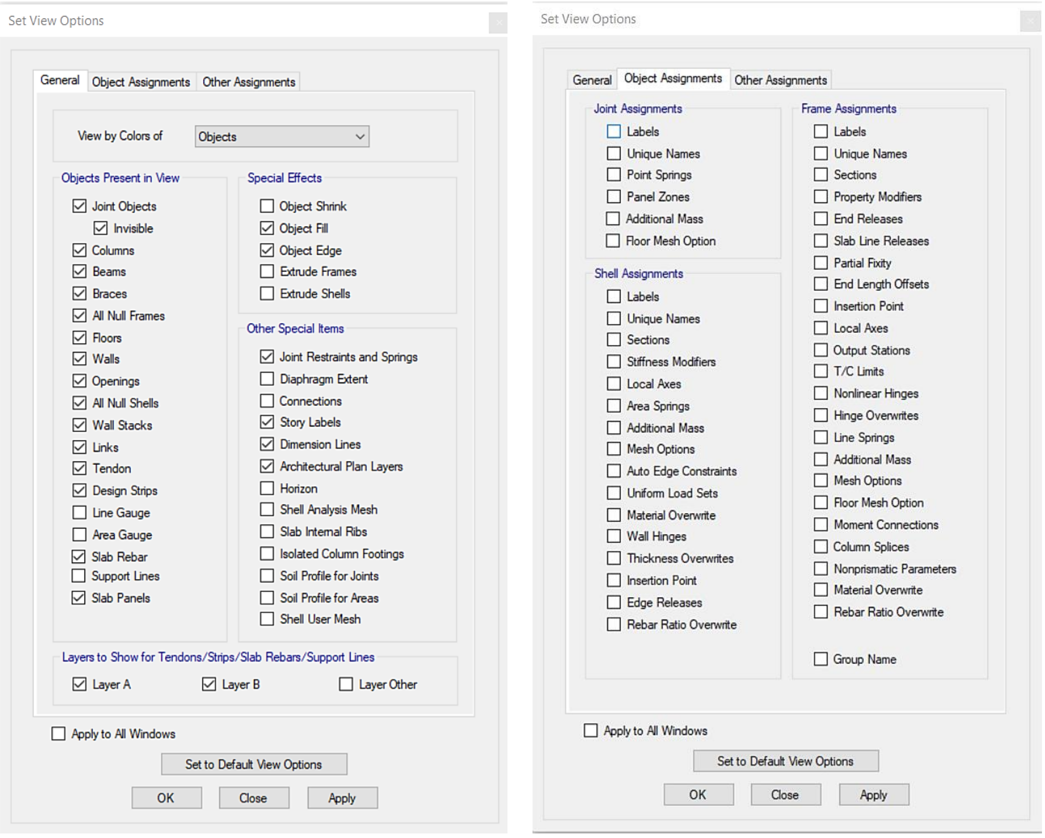


Figure 3.20: window to set view option

Step N° 04: Definition of Load Cases

For Etabs, each loading case must be defined in a general way. The loads are then applied to the appropriate nodes and elements and associated with an existing load case. The example discussed here includes the following loading cases:

- Linear static (us like; dead and live load)
- Nonlinear static (for pushover analysis)
- Nonlinear staged construction
- Modal
- Response spectrum (exp: Ex and Ey seismic loads in the case of the equivalent static method)
- Time history
- Buckling
- hyperstatic

Case of Static Loads (Dead and Live)

Select Static Load Cases from the Define menu. This option allows you to create the desired static loading cases.

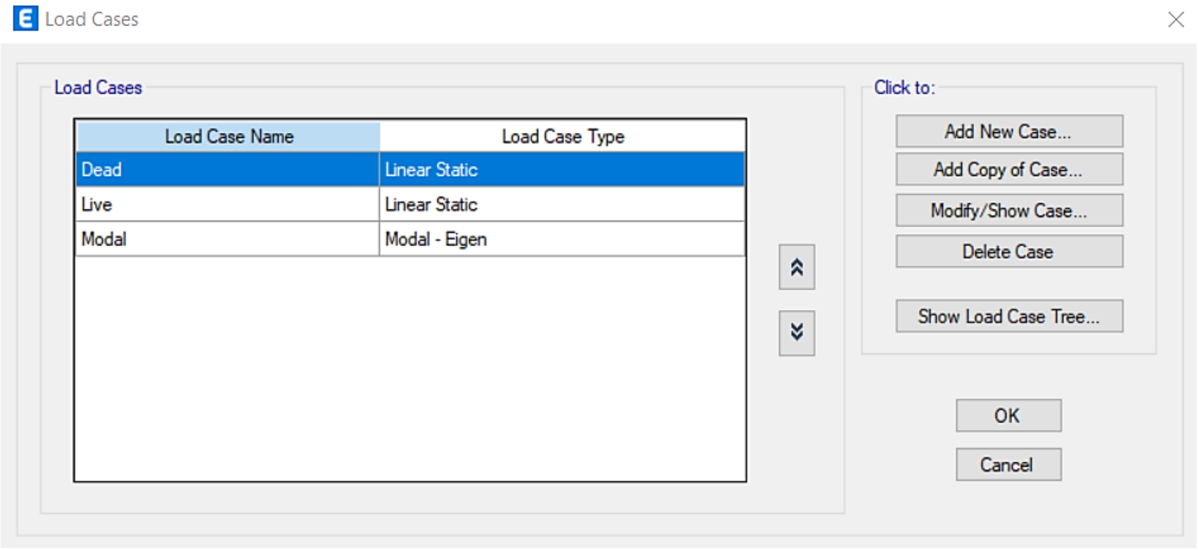


Figure 3.21: window for defining load cases in Etabs 20.

In the case of permanent loading G, if the self-weight multiplier is taken to be equal to 1, the self-weight of the components of the structure will be measured automatically by the software in the calculation. Otherwise, the computer software neglects the proper weight of the structural elements.

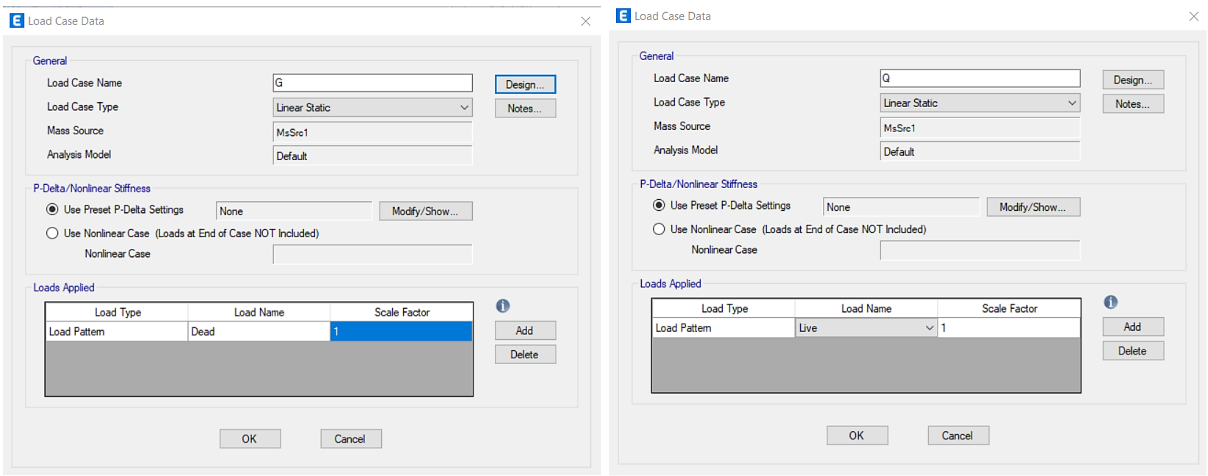


Figure 3.22: definition of the static load G and Q in Etabs 20.

Assignment of Static Loads

To assign the defined vertical static loads to the beams, we must first present the structure of the building in the XY plane, the first floor for example, on one of the displayed views of the model (3D and 2D) on screen. Click on the Clear icon in the floating toolbar to eliminate any selection.

Select in the chosen plan the beams to be loaded using the icon in the floating toolbar . Repeat this operation for beams with an equal load and belonging to the upper planes using the icon .

We assign the load with the ***Assign*** menu, then ***Frame Static Loads*** and ***Point and Uniform or Trapezoidal*** for a uniform and concentrated or trapezoidal load respectively.

By clicking on ***Assignment- Frame load-Distributed***, the following window appears, you must specify the name of the load case to be assigned (permanent load G for example), the type of the load (forces or moments), the direction of application of the load and the value of the load in Uniform Loads.

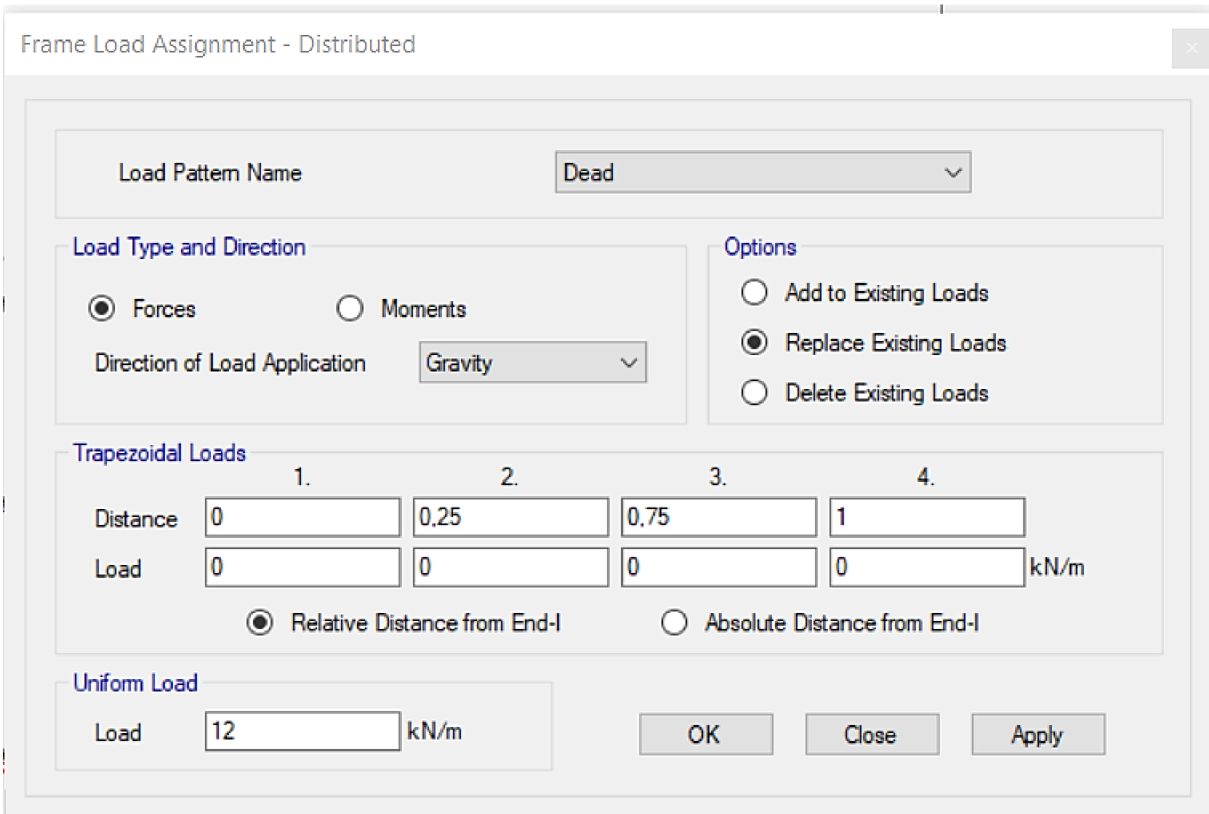


Figure 3.23: Assignment of distributed load to frame element in Etabs 20

Note: For a concentrated load in point (permanent or live), the position and the value of the load must be specified in Point Loads.

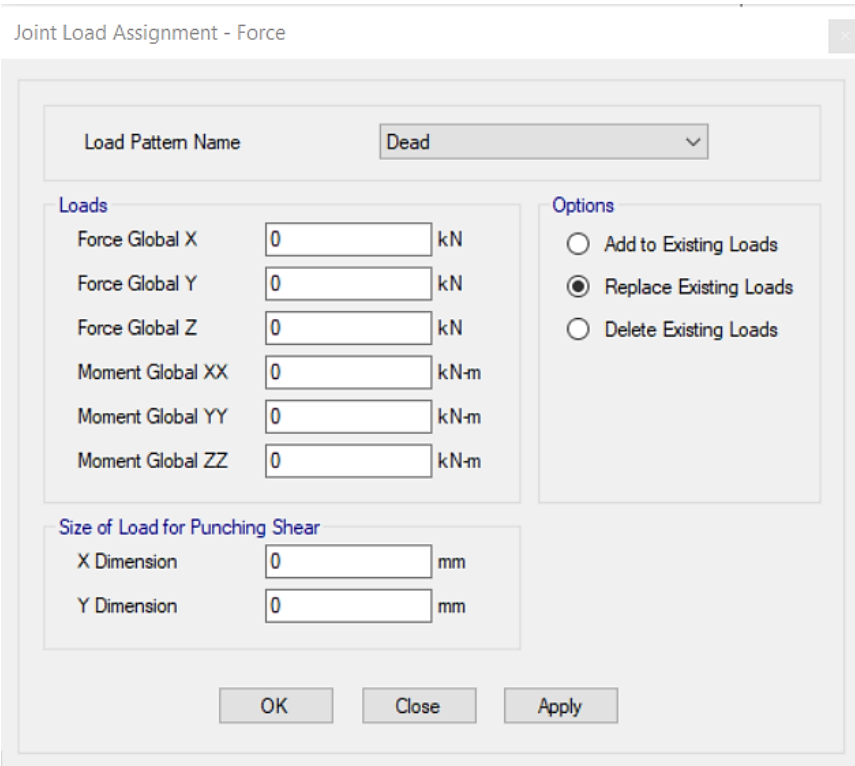


Figure 3.24: Assignment of point load to joint in Etabs 20

Manually Create User-Defined Load Combination

A load combination adds together or encompasses the findings of specific load instances. When conducting a linear analysis with overlaid findings, as 1.35*DL+1.5*LL, summation is frequently appropriate. Combining load patterns inside load cases is frequently the best approach for nonlinear analysis. Response envelopes can then be computed using load combinations. The outputs of a load combination comprise internal member forces and stresses as well as joint displacements and forces.

We enter the action combinations in the ***Define*** menu, then ***Loads combination*** and ***Add New Comb***. On the window that appears on the screen you must specify for the combination of the ultimate limit state, for example the following:

"ELS" Name of the action combination in ***Load Combination Name***, "***ADD***" Type of the action combination in ***Load Combination Load***, "ELU" Title of the combination, "1.35 and 1.5" weighting factors for loading cases "G and P" in ***Define Combination***.

When you type "OK" the name of the entered combination appears in the Define Load window Combinations.

Modify/Show Combination button

By using this button, you can change an already-existing combo without creating a new one.

- In the Load Combinations display box, select the load combination that has to be changed.
- To open the Load Combinations form, click the Modify/Show Combo button.
- Using the form's choices, make the appropriate changes.
- To go back to the Load Combinations form, click the OK button. Then, to close that form, click the OK button.

The button to add default design combos

1) The Add Default Design Combinations form will appear once you click the Add Default Design Combos button.
2) Choose the style of design (e.g., cold-formed steel frame, aluminum frame, concrete frame, steel frame).
3) After selecting the code using the Design menu > Design > View/Revise Preferences command, the application will add load combinations based on the code chosen. The code-generated combinations will be displayed in the Load Combinations display list, and the Load Combinations form will be displayed again.

Note: When using the ***Add Default Design Combos*** button or the Convert Combos to Nonlinear Cases button, multiple combinations can be selected in the Combinations display list using standard Windows selection techniques (e.g., drag the mouse over multiple combinations in the list; hold down the Ctrl key on the keyboard and click the mouse to make "random" selections).

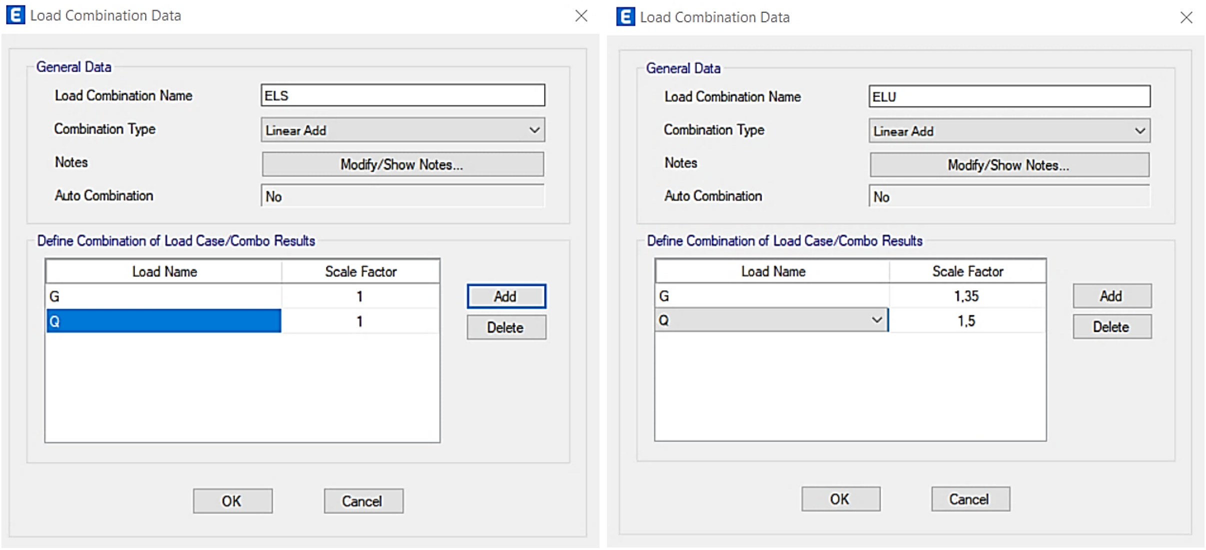


Figure 3.25: Definition of load combinations in Etabs (ELU and ELS)

Note: The "ADD" instruction represents the algebraic sum of the actions to be combined.

Step N° 05: Boundary Conditions

This step consists in specifying the boundary conditions (supports, etc ...) for the structure to be designed.

Supports (Restraints)

The template used has already placed double supports at the bottom of each pop-up. Since in this case we are talking about recesses, they must be modified.

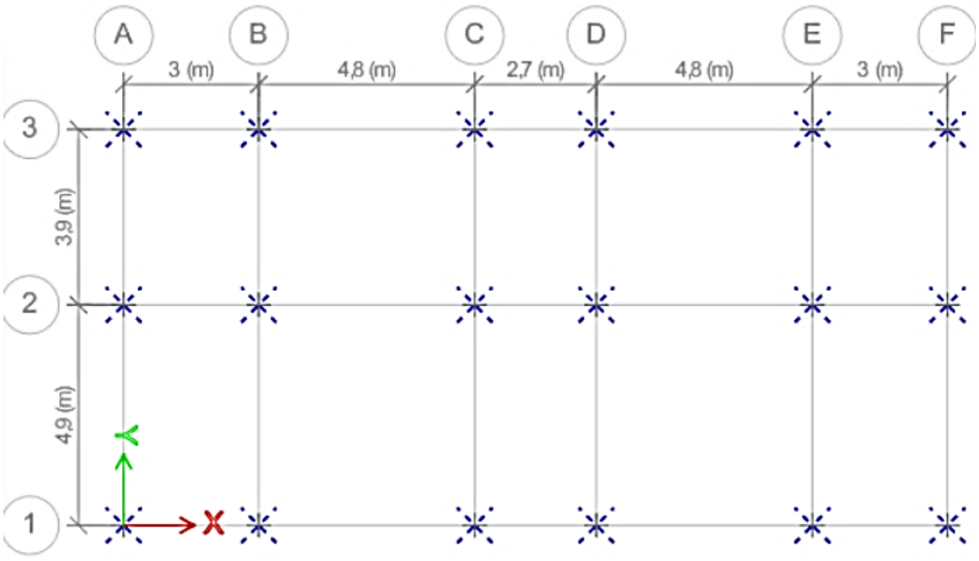


Figure 3.26: Selection of support points for the attributes the support conditions

To acquire the Joint Assignment - Restraints form, click the **Assign menu** > **Joint** > **Restraints** command.

Restraints in Global Directions options: These check boxes identify the six possible degrees of freedom available for a joint object. When a check box is checked, that degree of freedom is restrained. When a check box is unchecked, that degree of freedom will be unrestrained, assuming that the degree of freedom has been designated as active for the model.

Fast Restraints options: Choose one of the following four buttons to quickly set the restraint conditions. Note that when one of the buttons is checked, the check boxes in the Restraints in Global Directions area of the form become checked or unchecked, depending on the selection made.

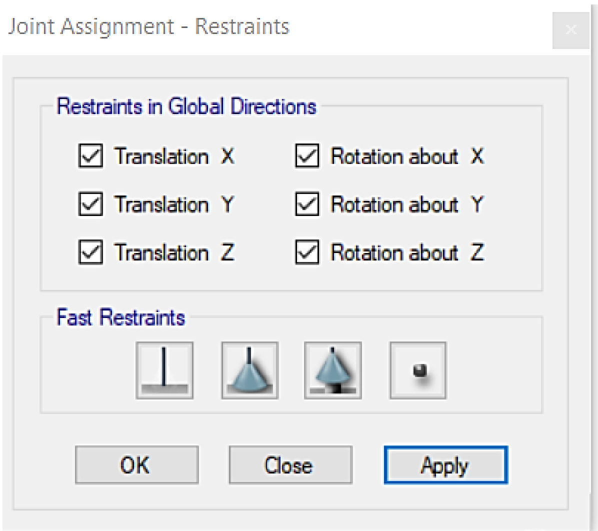


Figure 3.27: Types of supports available in Etabs 20

- Fixed base restraint button: Click this button to restrain all six degrees of freedom.
-  Pinned base button: Click this button to restrain all three translation degrees of freedom and to specify that all three rotation degrees of freedom unrestrained.
-  Roller support button: Press this button to limit the Z translation solely. There are no restrictions on any other degree of freedom.
-  No support button. To indicate that there are no restrictions on any degree of freedom, click this button.

Click the Apply button to apply the assignment and leave the form open to allow assignment to another set of selected objects; when all assignments have been made, click the Close button to close the form. When the Apply button is used, the form will remain open until it is closed by clicking the Close button.

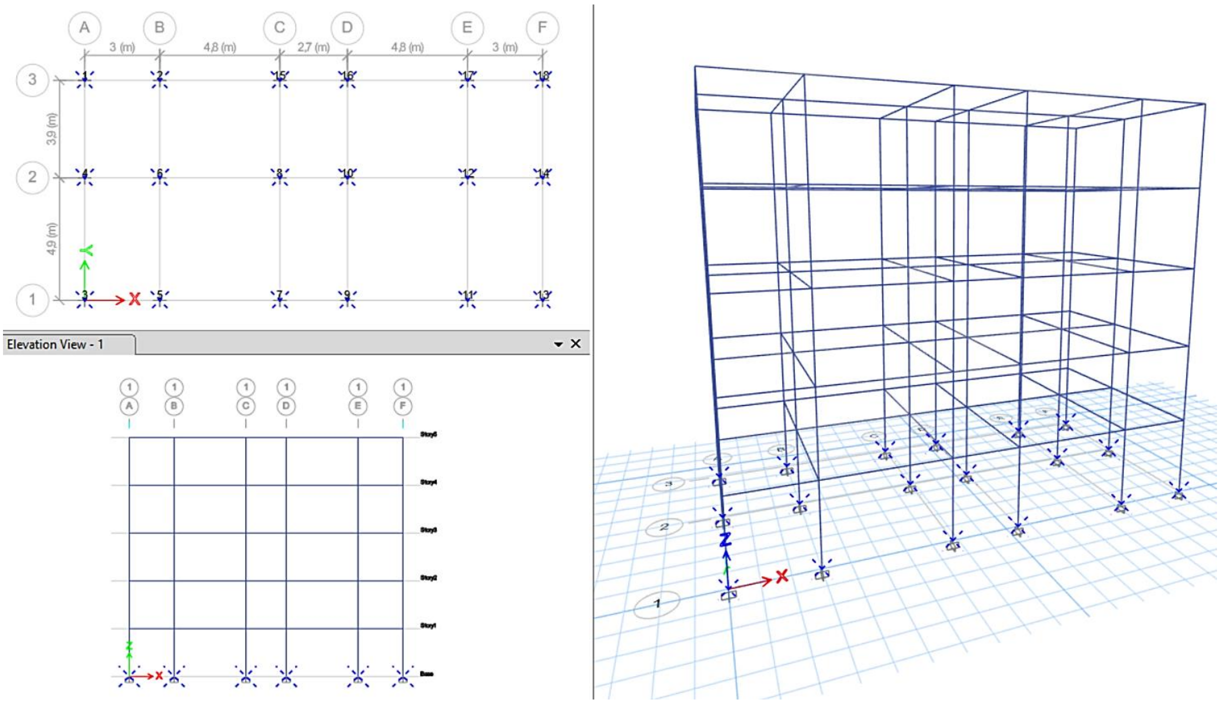


Figure 3.28: Change from double supports to fixed supports.

Creating the Mass Source

To access the Define Mass Source form, use the **Define menu** > **Mass Source** command. Select whether a building's mass comes from loads, from self-mass and additional mass, or from a mix of loads, additional mass, and self-mass. Additionally, indicate the location and consideration of lateral mass.

This are of the form has the following options:

- **Element Self Mass option**: A mass per unit volume is one of the parameters listed in the material properties that are connected to each structural element. If this check box is selected, each structural element's volume multiplied by its chosen mass per unit volume is how ETABS calculates the building mass related to the element mass. This is how things are by default. Additionally, the mass and weight as stated in the link property description are included in ETABS if the link properties have been applied to a joint or frame object.

- ***Additional Mass option***: Additional mass can also be assigned to accommodate cladding, partitions, and other features. Refer to the Additional Line Mass, Additional Area Mass, and Additional Point Mass sections. In all three translational directions, the extra mass is applied to every joint in the structure based on its tributary area. Keep in mind that you can enter more masses as negative values.
- ***Specified Loads Patterns Option***: Indicate a load pattern combination that provides a basic definition of the structure's mass. By dividing the load combination by the gravitational multiplier, g, the mass is equal to the weight that exists. Measuring the mass only takes into account the global Z-direction loads. On a tributary area basis, this mass is applied in all three translational directions to every joint in the structure. Positive masses are defined as net downward loads applied to a joint. At a joint, the mass is 0 if the net load acting on the joint is upward. In the ETABS, negative mass is prohibited. Every joint in the structure receives the mass by default in all three translational directions on a tributary area basis.
- ***Adjust Diaphragm lateral Mass*** to ***Move Mass Centroid*** option. Use this option to move the mass centroid associated with the lateral diaphragm.

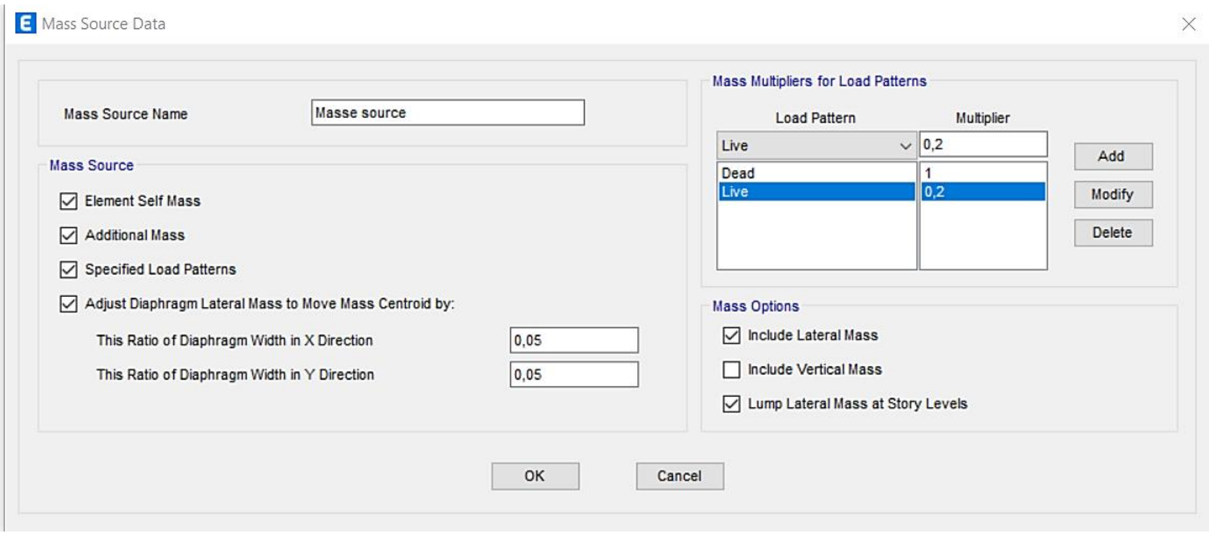


Figure 3.29: Creating Mass source and option definition

Define Constraints (Diaphragms)

constraints make it possible to link degrees of freedom (DDL) of one or more nodes to the degrees of freedom of a single node. This has the effect of reducing the number of equations to be solved.

To view the Define Diaphragms form, click the Define menu > Diaphragms command. The names of all the diaphragms that are currently defined are displayed in the Diaphragms display list.

The diaphragm definition can be added, changed, or removed by selecting one of the following buttons:

Add New Diaphragm

1) To open the Diaphragm Data form, click the Add New Diaphragm button.
2) Type a different name into the Diaphragm edit box, or use the default name that is offered.
3) Select the Rigid or Semi Rigid option to indicate the diaphragm's rigidity. These choices alone impact the model's analysis. A totally rigid diaphragm is presumed if the rigid option is chosen. The stiffness of the components that make up the diaphragm determines the diaphragm's in-plane rigidity if the semi-rigid option is chosen.
4) Press the OK.

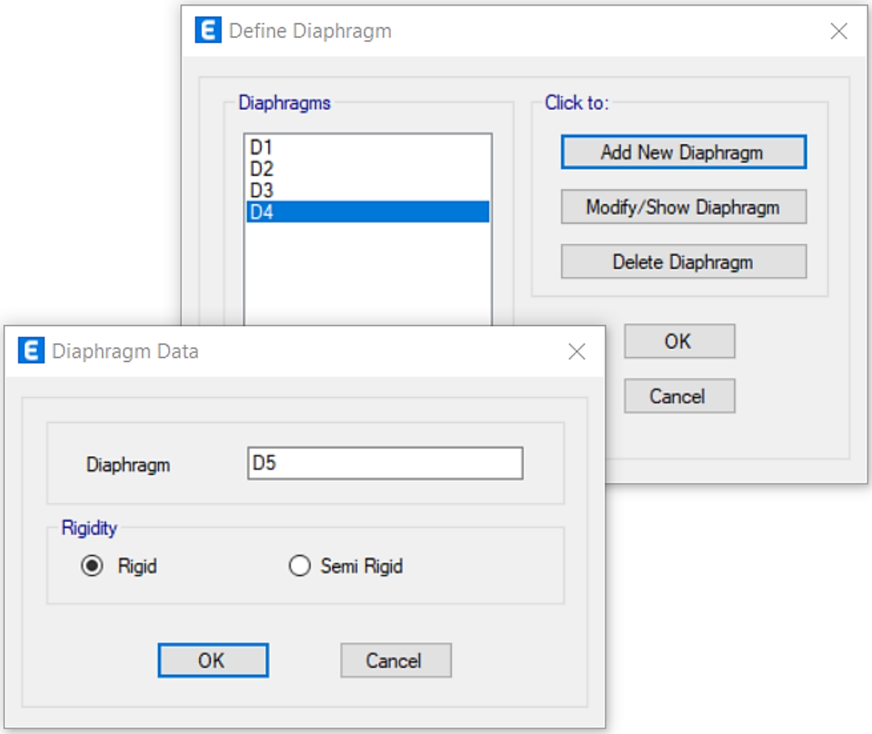


Figure 3.30: Diaphragm Data window

Note: It is necessary to assign a name for the Add Diaphragm assigned to the first floor and specify that the degree of freedom will be common to all the nodes of this floor. Choose D1 as the name and select the rigidity type of diaphragm (Rigid or Semi Rigid), on the Diaphragm Data window.

Click on "OK" and repeat the operation for all the floors by selecting a different name (D2, D3, D4, and D5).

Joint Assignments - Diaphragms

Use the ***Assign*** menu > ***Joint*** > ***Diaphragms*** command to assign previously defined diaphragms to selected joint objects.

1. Select the joint(s) to be assigned a diaphragm. A plane parallel to the global X-Y plane is usually where the chosen joints should all be located. The diaphragm only includes the joints that were genuinely chosen for inclusion.
2. Click the ***Assign*** menu > ***Joint*** > ***Diaphragms*** command to access the Joint Assignments - Diaphragms form.
3. Select a diaphragm property from the display list and click the ***Apply*** button to make the assignment. Any previously assigned diaphragm properties will be replaced by this assignment.

4.

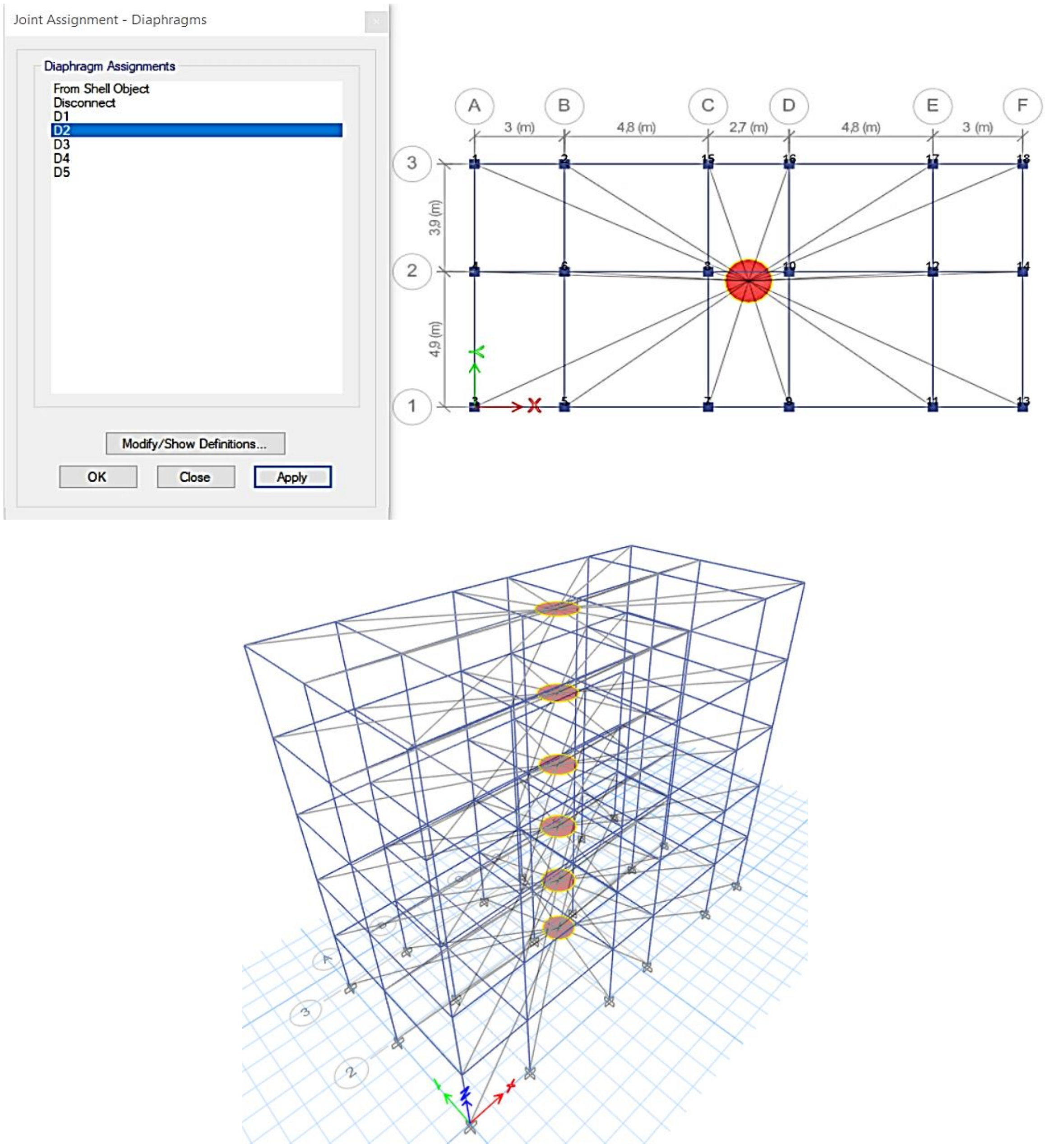


Figure 3.31: Joint Assignments (Diaphragms) in Etabs 20

Step N° 06: Run Analysis

Before starting the execution of the calculation example, it must be specified that the analysis will be performed in three dimensions.

1) Click the ***Analyze*** menu > ***Set Active Degrees of Freedom*** command to access the ***Active Degrees of Freedom*** form.

2) Use the form to specify the Building Active Degrees of Freedom: The possible degrees of freedom for a building are UX, UY, UZ, RX, RY and RZ. When a degree of freedom check box is checked, that degree of freedom is active for the model. Alternatively, the following four buttons can be used to quickly set the degrees of freedom for the typical cases that might arise.
3) Full 3D button: Enables each of the six degrees of freedom. For the great majority of building models, this option should be used.
4) XZ Plane button: Enables the degrees of freedom for RY, UX, and UZ. It is designed for global XZ plane-modeled two-dimensional frames.
5) YZ Plane button: This button activates the RX, UZ, and UY degrees of freedom. It is designed for global YZ plane-modeled two-dimensional frames.
6) No Z Rotation button: Activates every degree of freedom but not RZ.

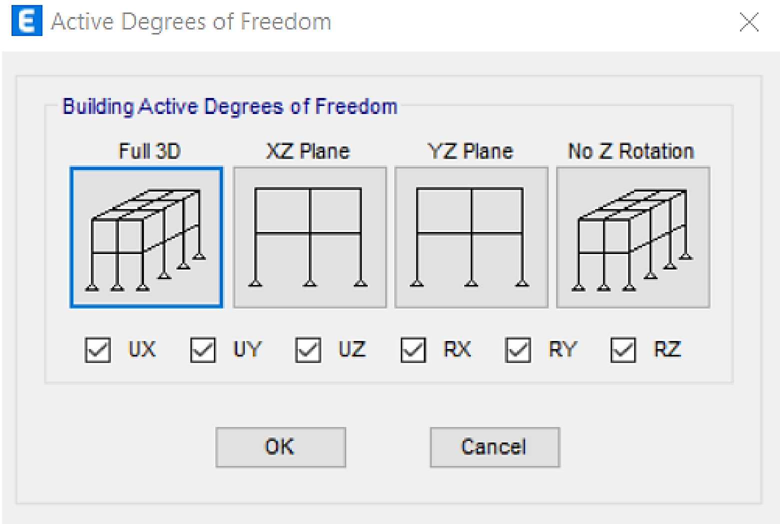
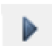


Figure 3.32: Analysis sept of Building Active Degrees of Freedom

The execution of the problem can be started by selecting Analyze and Run Analysis. It can also be started by pressing F5 or the icon . If there are any errors, you can return to the graphics module and make the necessary corrections.

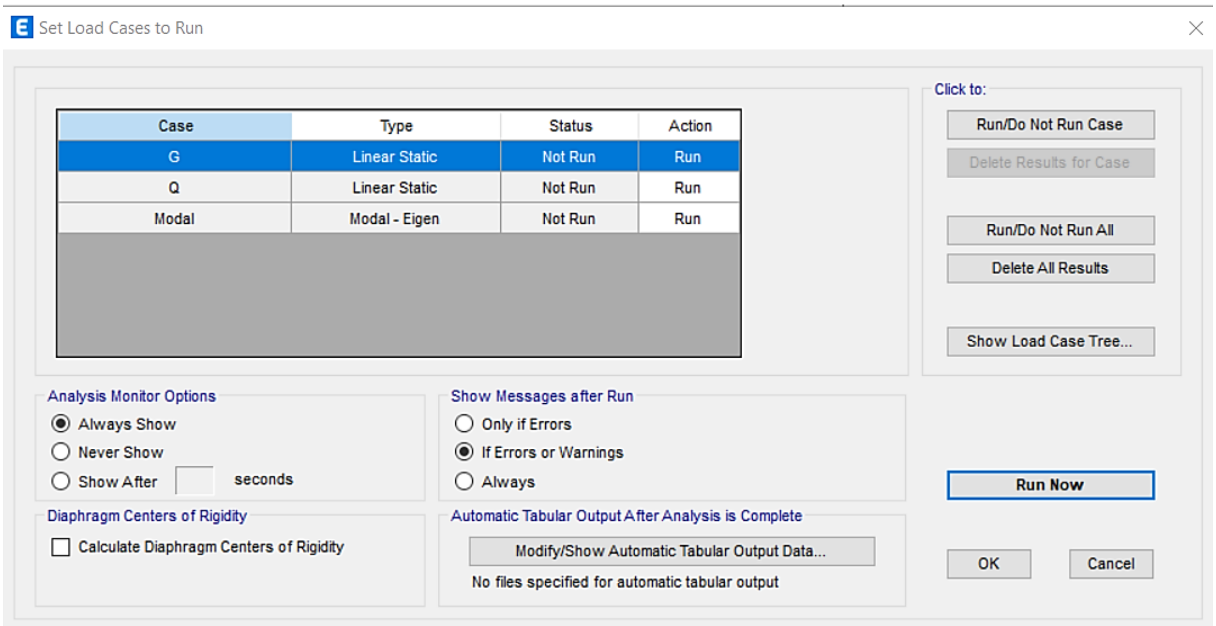


Figure 3.33: Menu of select load cases analysis to run.

Check the model before the analysis

Before performing an analysis, check your structure using the Check Model form, which can be accessed via the Analyze menu > Check Model command.

- ✓ **Joint Checks** - Joints/Joints within Tolerance, Joints/Frames within Tolerance, Joints/Shells within Tolerance: Check the appropriate checkbox to specify that the program check the alignment of joint, frame, and shell objects. The program will display a message box stating that errors have not been found, or identifying any locations in the model where joints are outside the tolerance distance as specified in the Length Tolerance for checks edit box on the form, such that the overlaps or gaps will affect analysis results.
- ✓ **Frame Checks** - Frame Overlaps, Frame Intersections within Tolerance, Frame Intersections with Area Edges: Check the appropriate checkboxes to specify that the program verify where frames overlap and intersect. The program will display a message box stating that errors have not been found, or identifying any locations in the model where frames overlap or frames do not adequately intersect, as specified by the distance in the Length Tolerance for checks edit box on the form, such that the overlaps or gaps will affect analysis results.
- ✓ **Check Loading for All Stories**: When this check box is checked, the program will check loading on a story-by-story basis. The program will display a message box stating that errors have not been found, or identifying loading that may affect analysis results.

- ✓ **Trimor Extend Frames and Move Joints to Fix Problems:** When this box is checked, the Check Model command will move joints (including the ends of frame elements) to fix connectivity, over laps or such. The distance moved in any one direction will not exceed the Length Tolerance listed above.
- ✓ **Joint Story Assignment:** When this box is checked, the Check Model command will correct any story assignments for joints that should belong to a different story.

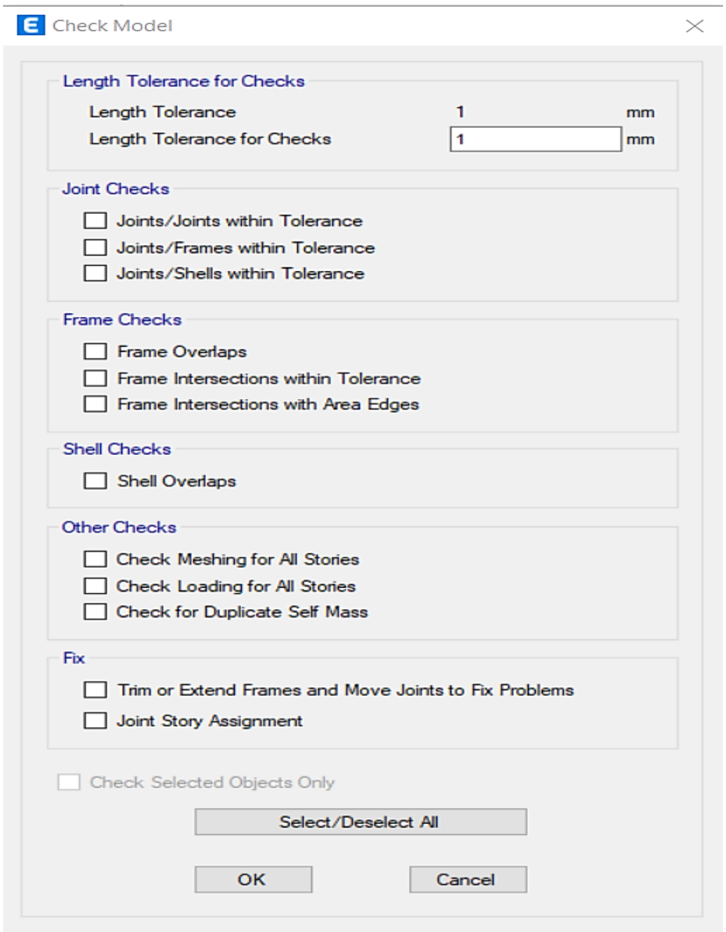


Figure 3.34: The model checking options before launching the analyses.

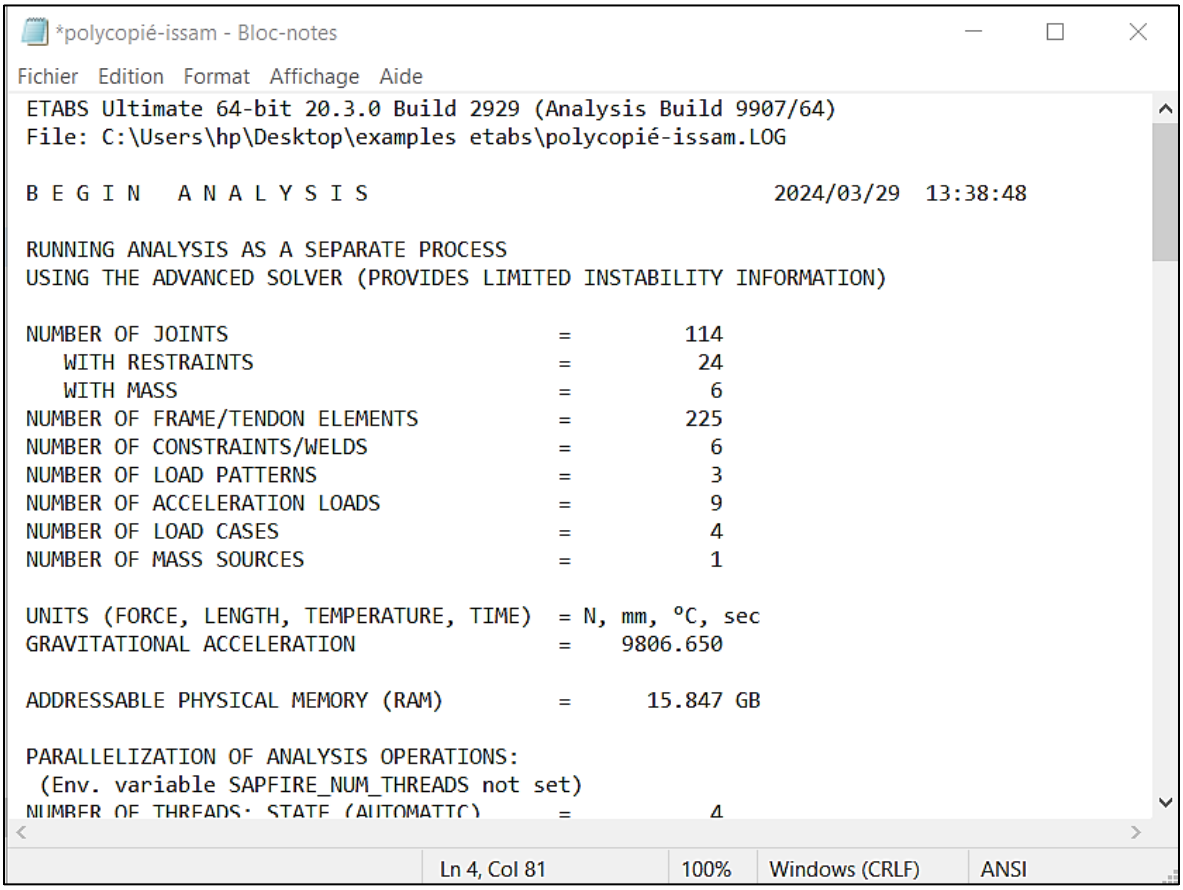
Step N °07: Visualization and Exploitation of the Results

After executing the model, the Etabs builds three extension files (.ebk, edb., .OUT, and .LOG).

The extension file (.LOG) makes it possible to obtain :

- ✓ number of frame/tendon elements
- ✓ total number of equilibrium equations
- ✓ forming stiffness at zero (unstressed) initial conditions

- ✓ number of mass degrees of freedom
- ✓ number of eigen modes found
- ✓ number of iterations performed


*polycopié-issam - Bloc-notes

Fichier Edition Format Affichage Aide

```
ETABS Ultimate 64-bit 20.3.0 Build 2929 (Analysis Build 9907/64)
File: C:\Users\hp\Desktop\examples etabs\polycopié-issam.LOG

B E G I N   A N A L Y S I S                        2024/03/29  13:38:48

RUNNING ANALYSIS AS A SEPARATE PROCESS
USING THE ADVANCED SOLVER (PROVIDES LIMITED INSTABILITY INFORMATION)

NUMBER OF JOINTS                             =         114
   WITH RESTRAINTS                           =          24
   WITH MASS                                 =           6
NUMBER OF FRAME/TENDON ELEMENTS              =         225
NUMBER OF CONSTRAINTS/WELDS                  =           6
NUMBER OF LOAD PATTERNS                      =           3
NUMBER OF ACCELERATION LOADS                 =           9
NUMBER OF LOAD CASES                         =           4
NUMBER OF MASS SOURCES                       =           1

UNITS (FORCE, LENGTH, TEMPERATURE, TIME)  = N, mm, °C, sec
GRAVITATIONAL ACCELERATION                   =     9806.650

ADDRESSABLE PHYSICAL MEMORY (RAM)            =      15.847 GB

PARALLELIZATION OF ANALYSIS OPERATIONS:
 (Env. variable SAPFIRE_NUM_THREADS not set)
NUMBER OF THREADS: STATE (AUTOMATIC)         =           4
```

Ln 4, Col 81 | 100% | Windows (CRLF) | ANSI

Figure 3.35: LOG file obtains after analysis finish.

Visualization of diagrams result:

Click the *Display menu* > *Forces/Stresses Diagram* > *Frames* to display the following four types of diagrams for the beam.

- ➢ Applied loads
- ➢ Shear diagram
- ➢ Moment diagram
- ➢ Deflection diagram

The Display menu allows you to display the internal forces with Show Element Forces /Stresses and Frames. Select on the window that appears on your screen the combination of load and the desired effort in Load and component respectively.

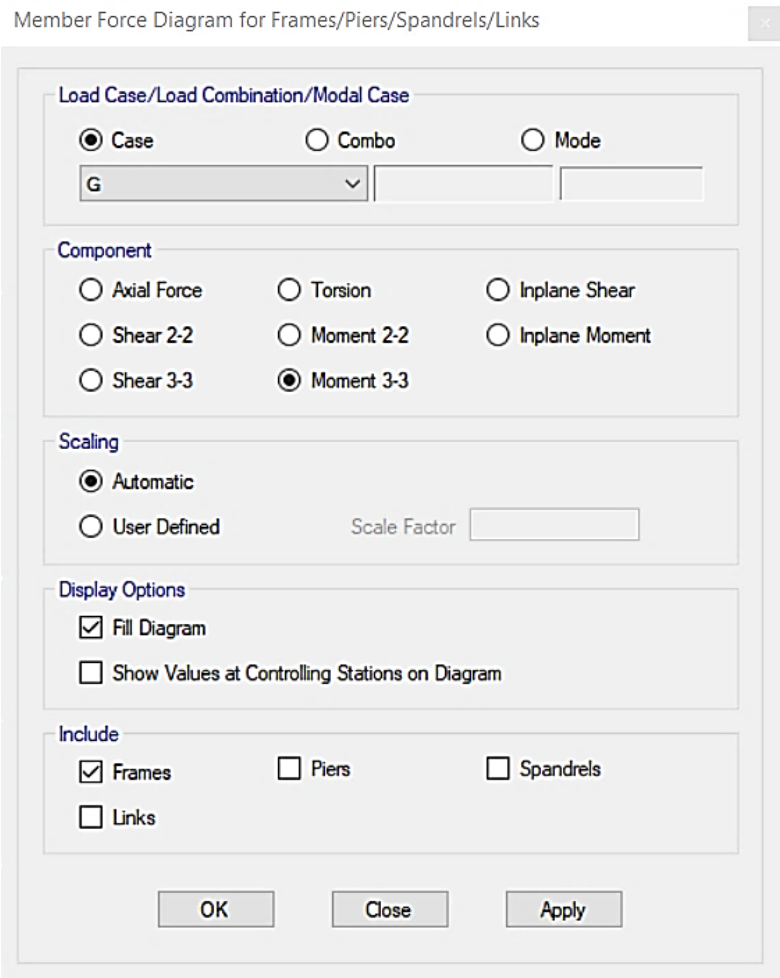


Figure 3.36: Menu of force, moment, shear diagrams and display options.

Load Case/Load Combination / Modal Case: For the purpose of displaying the frame, pier, spandrel, or link forces, select the load case, load combination, or modal case using the radio buttons and drop-down lists provided.

Note: Plotting of member forces is possible for several scenarios, including time history, static nonlinear, response spectrum, static load, and load combination cases. Establish a time for the forces' display in time history scenarios as well. Indicate a step for the forces' display in static nonlinear scenarios as well.

Component options: Indicate which aspect of force to exhibit. Select one by one. Only when a 2-D view is the currently active window (the one where the forces will be shown) are the in-plane moment and shear items available. When the local 2-axis of the columns is in the plane of the frame, this view is helpful for showing two-dimensional frames. The force that is shown when the in-plane option is selected is composed of the relevant components from the local 2 and 3 axes if the member's local axes are rotated so that neither the local 2 nor the local 3 axes are in the 2-D plane.

Display Options: Use these options to specify how the diagram is displayed.

- ***Fill Diagram*** check box: When this check box is checked, the force diagrams will be displayed filled with no text values. Note that if the Show Values on Diagram check box is checked, first uncheck it before checking the Fill Diagram check box.
- ***Show Values at Controlling Stations on Diagram*** check box: When this check box is checked, the force diagrams will be displayed with text values and unfilled. Note that if the Fill Diagram checkbox is checked, first uncheck it before checking the ***Show Values*** at ***Controlling Stations*** on ***Diagram*** check box.
- Leave both check boxes unchecked to display the form diagrams unfilled with no text values.

Internal Forces: The Display menu allows you to display the internal forces with Show Element Forces /Stresses and Frames. Select on the window that appears on your screen the combination of load and the desired effort in Load and component respectively.

The ***Axial Force*** option displays the diagram of the normal forces. The Shear 2-2 and 3-3 option makes it possible to display the diagram of the cutting forces in the directions 2-2 and 3-3. The Moment 3-3 and 2-2 option displays the diagram of the bending moments in the directions 3-3 and 2-2. The following figures show the distribution of the normal force for the ELU combination.

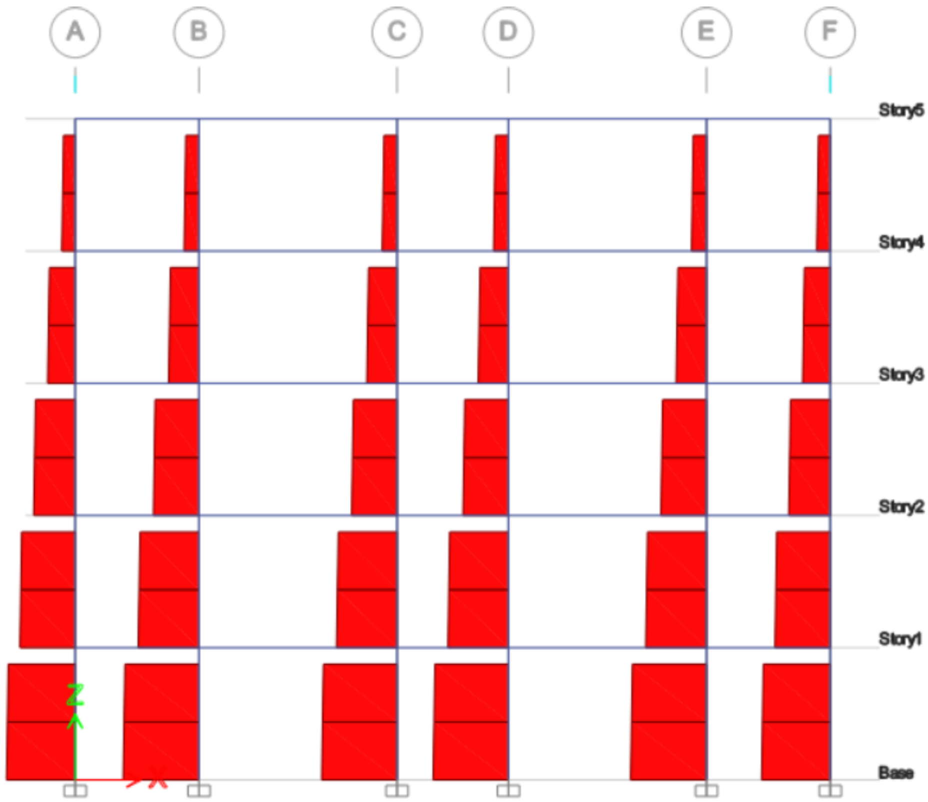


Figure 3.37: Elevation view of axial force diagram for one frame.

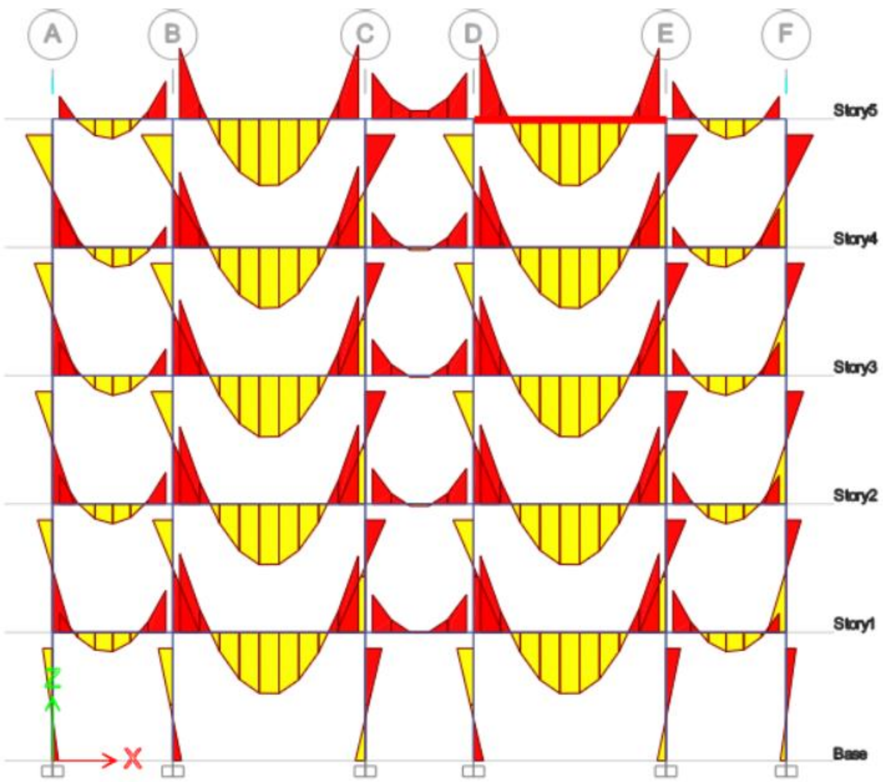


Figure 3.38: Full diagram of moment 3-3 for one frame.

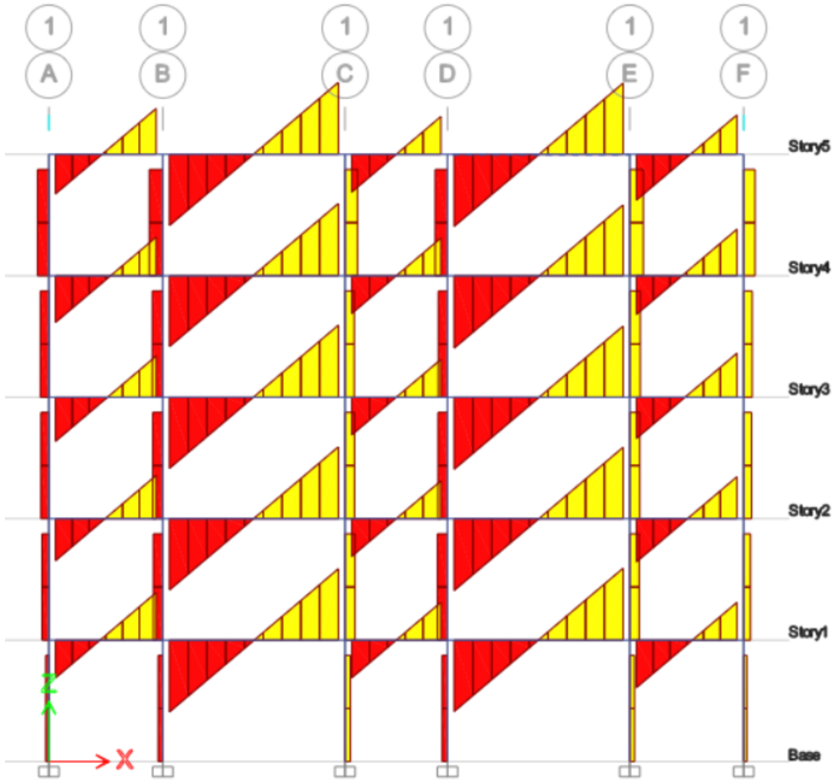


Figure 3.39: Full diagram of shear 2-2.

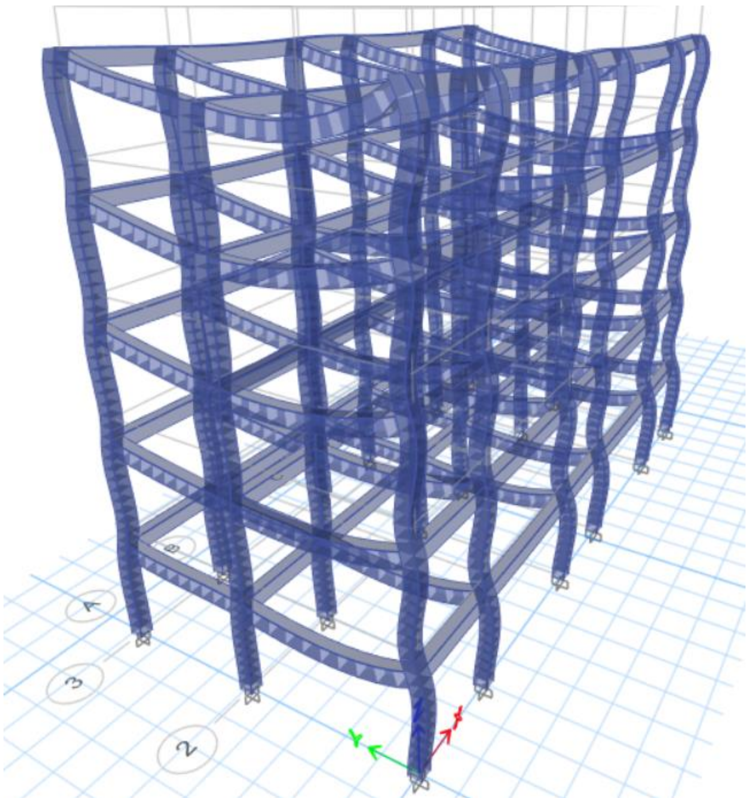


Figure 3.33: Deformed shape of study model in Etabs.

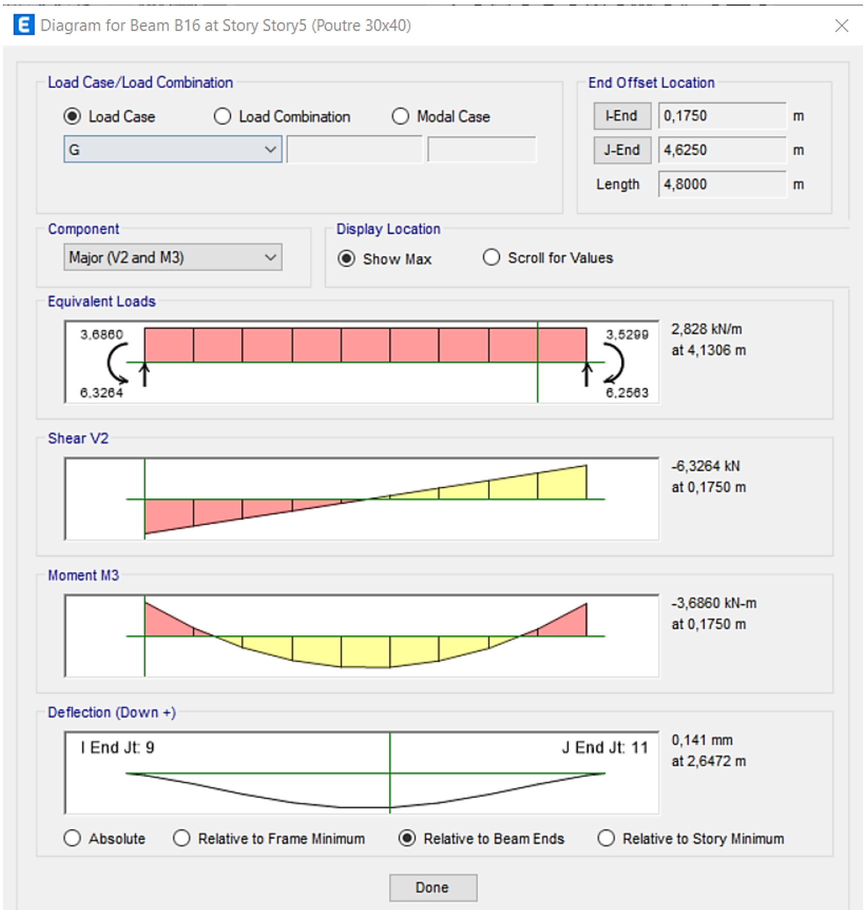


Figure 3.40: Values at Controlling Stations on Diagram.

Support (restraints)

The Display menu allows you to display the reactions with the Show Element Strengths/Stresses and Joints option. Select Reactions and the desired load case. The reactions will be displayed on the supports. You can print this view with the File menu and Print Graphics.

Figure 3.41: Support reaction values and direction display in Etabs version 20.3.

Choose Tables for Editing, Display, Printing, and Export

Use the Choose Data Tables form to select the types of model data to be edited interactively, displayed, printed, exported or selected. The options available on the form depend on the command used to access the form.

The tables available for interactive editing, display, printing, export or selection are listed in follow figure. Check the check box associated with an item to include that item in the editing, display, print, export or selection.

This menu can be used to do the following:

- Set Output Options: This command is identical to the Modify/ Show Options button as described below.
- Set Output Selections: This is only available when Sections Cuts have been defined and results are currently available. It allows you to select between the various defined section cuts.
- Select Load Patterns: This command is identical to the Select Load Patterns button as described below.
- Select Load Cases: This command is identical to the Select Load Cases button as described below.
- Show Named Set: This command is identical to the Show Names Set button as described below.
- Set Table File Format: This command is identical to the Table File Format button as described below.
- Display various output tables: See the topic on Joint Drift Output Tables for more information on how to display similar tables.

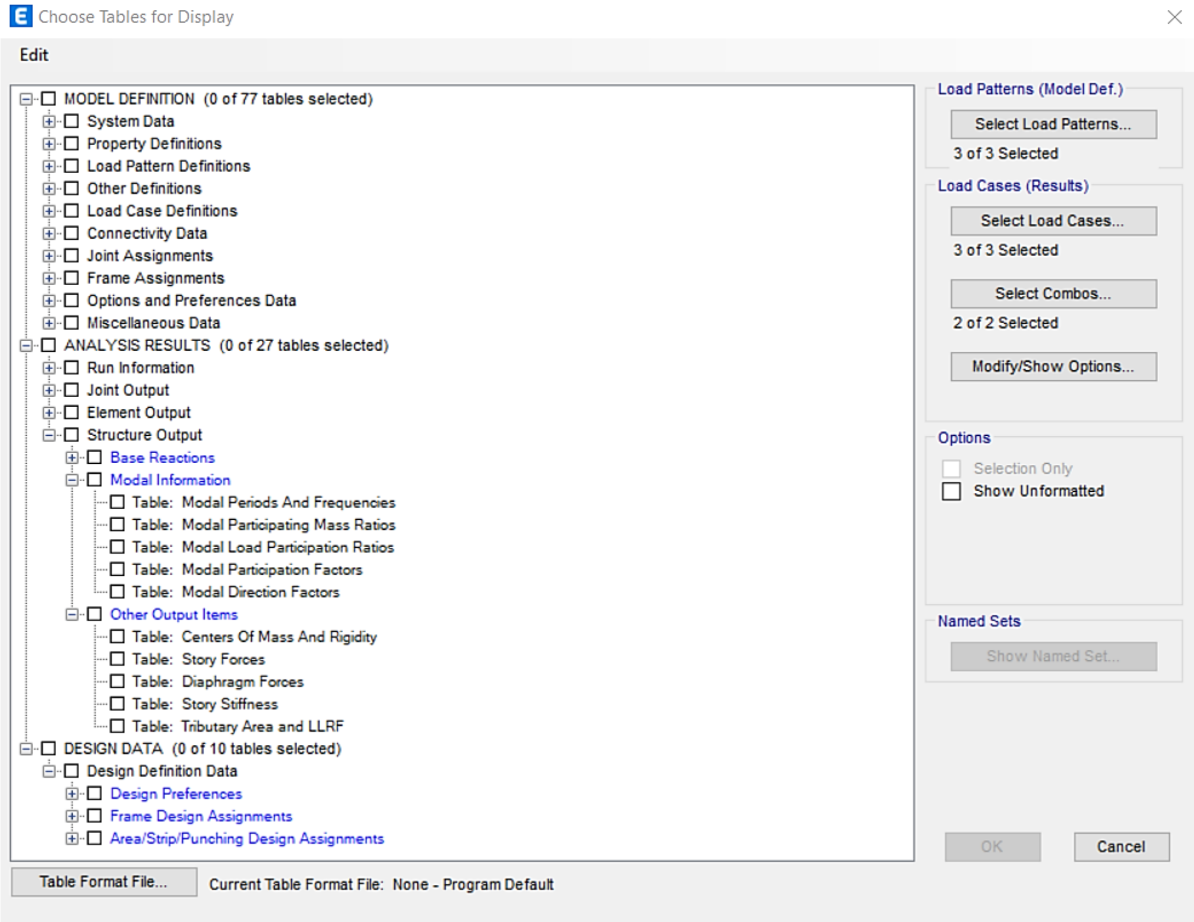
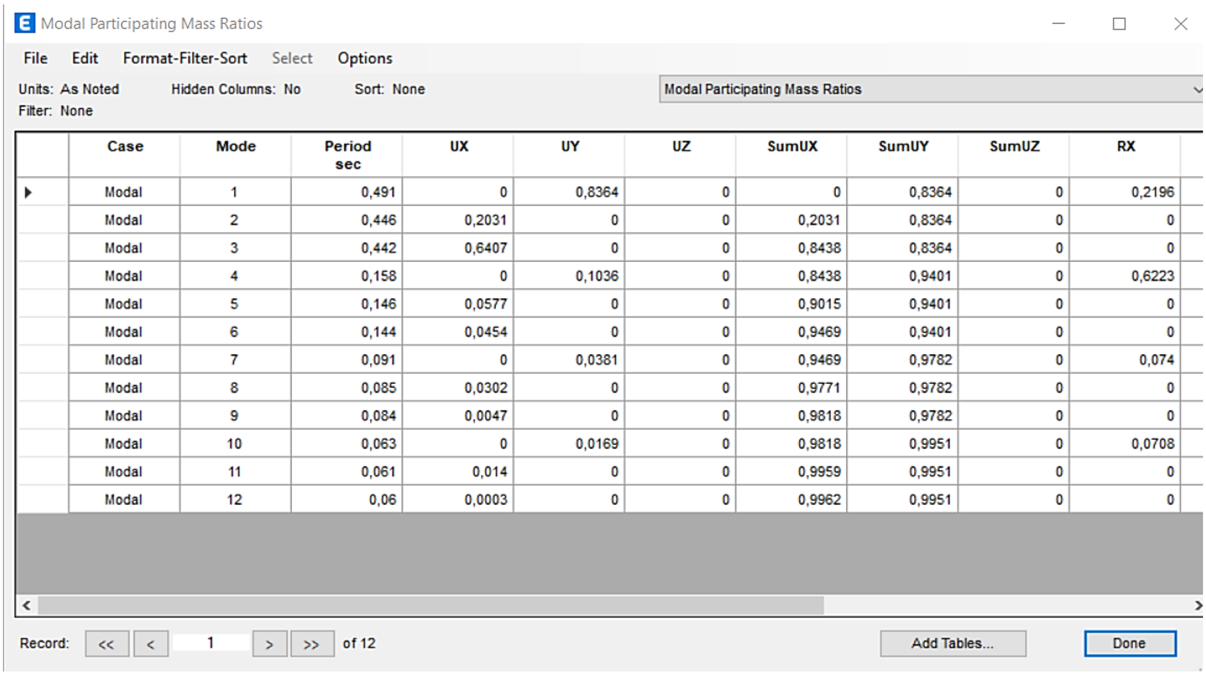


Figure 3.42: Tables for Editing, Display, Printing, and Export results in Etabs 20.



Case	Mode	Period sec	UX	UY	UZ	SumUX	SumUY	SumUZ	RX
Modal	1	0,491	0	0,8364	0	0	0,8364	0	0,2196
Modal	2	0,446	0,2031	0	0	0,2031	0,8364	0	0
Modal	3	0,442	0,6407	0	0	0,8438	0,8364	0	0
Modal	4	0,158	0	0,1036	0	0,8438	0,9401	0	0,6223
Modal	5	0,146	0,0577	0	0	0,9015	0,9401	0	0
Modal	6	0,144	0,0454	0	0	0,9469	0,9401	0	0
Modal	7	0,091	0	0,0381	0	0,9469	0,9782	0	0,074
Modal	8	0,085	0,0302	0	0	0,9771	0,9782	0	0
Modal	9	0,084	0,0047	0	0	0,9818	0,9782	0	0
Modal	10	0,063	0	0,0169	0	0,9818	0,9951	0	0,0708
Modal	11	0,061	0,014	0	0	0,9959	0,9951	0	0
Modal	12	0,06	0,0003	0	0	0,9962	0,9951	0	0

Figure 3.43: view of modal participation mass result in Etabs version 20.

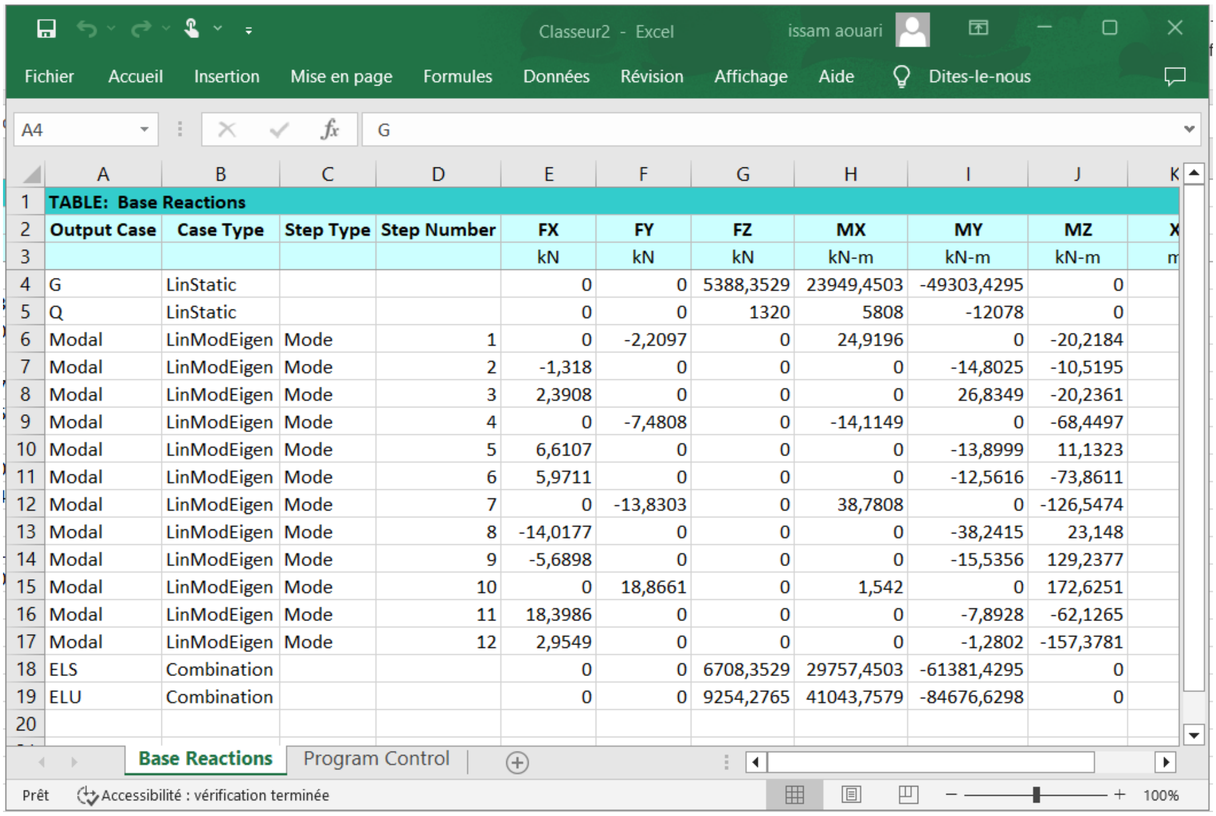


TABLE: Base Reactions									
Output Case	Case Type	Step Type	Step Number	FX	FY	FZ	MX	MY	MZ
				kN	kN	kN	kN-m	kN-m	kN-m
G	LinStatic			0	0	5388,3529	23949,4503	-49303,4295	0
Q	LinStatic			0	0	1320	5808	-12078	0
Modal	LinModEigen	Mode	1	0	-2,2097	0	24,9196	0	-20,2184
Modal	LinModEigen	Mode	2	-1,318	0	0	0	-14,8025	-10,5195
Modal	LinModEigen	Mode	3	2,3908	0	0	0	26,8349	-20,2361
Modal	LinModEigen	Mode	4	0	-7,4808	0	-14,1149	0	-68,4497
Modal	LinModEigen	Mode	5	6,6107	0	0	0	-13,8999	11,1323
Modal	LinModEigen	Mode	6	5,9711	0	0	0	-12,5616	-73,8611
Modal	LinModEigen	Mode	7	0	-13,8303	0	38,7808	0	-126,5474
Modal	LinModEigen	Mode	8	-14,0177	0	0	0	-38,2415	23,148
Modal	LinModEigen	Mode	9	-5,6898	0	0	0	-15,5356	129,2377
Modal	LinModEigen	Mode	10	0	18,8661	0	1,542	0	172,6251
Modal	LinModEigen	Mode	11	18,3986	0	0	0	-7,8928	-62,1265
Modal	LinModEigen	Mode	12	2,9549	0	0	0	-1,2802	-157,3781
ELS	Combination			0	0	6708,3529	29757,4503	-61381,4295	0
ELU	Combination			0	0	9254,2765	41043,7579	-84676,6298	0

Figure 3.44: Output tables to office Excel software.

Saving tabular analysis findings automatically

After the analysis is complete, the following process can be used to automatically save the Etabs results:

1. **Define database table named sets**: To establish a named set for the database tables that will be exported, make the following selections:

   Define > Named Sets > Tables, as shown in follow Figure 3.45:

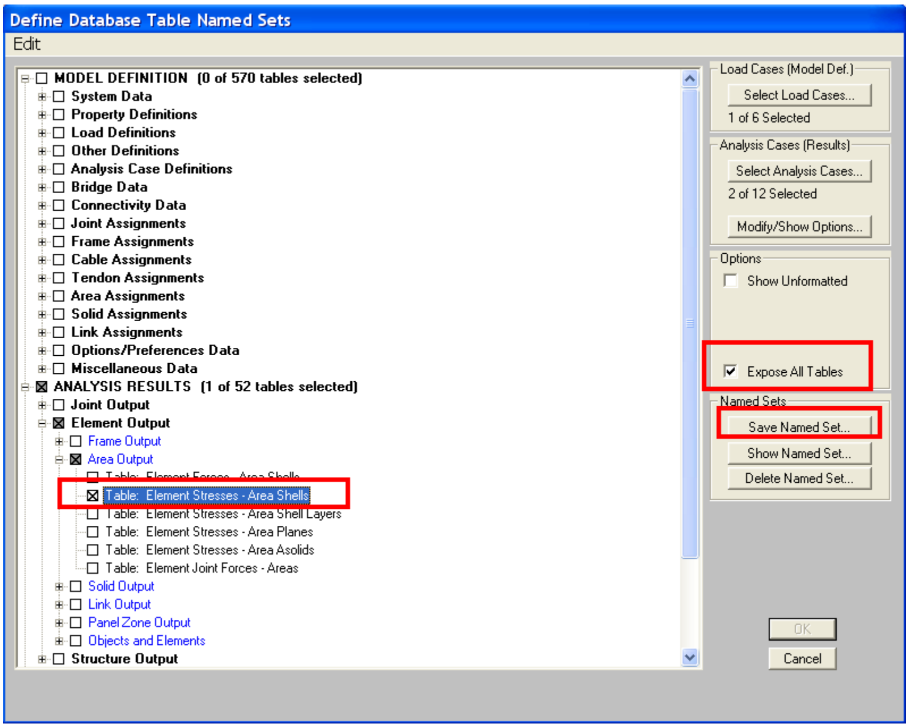


Figure 3.45: Menu how to export database tables by creating a named set.

2. Choose the output file: As seen in Figure 3.46 below, select output to a Tabular File from the Analysis Options menu, which is accessible by selecting Analyze > Set Analysis Options.

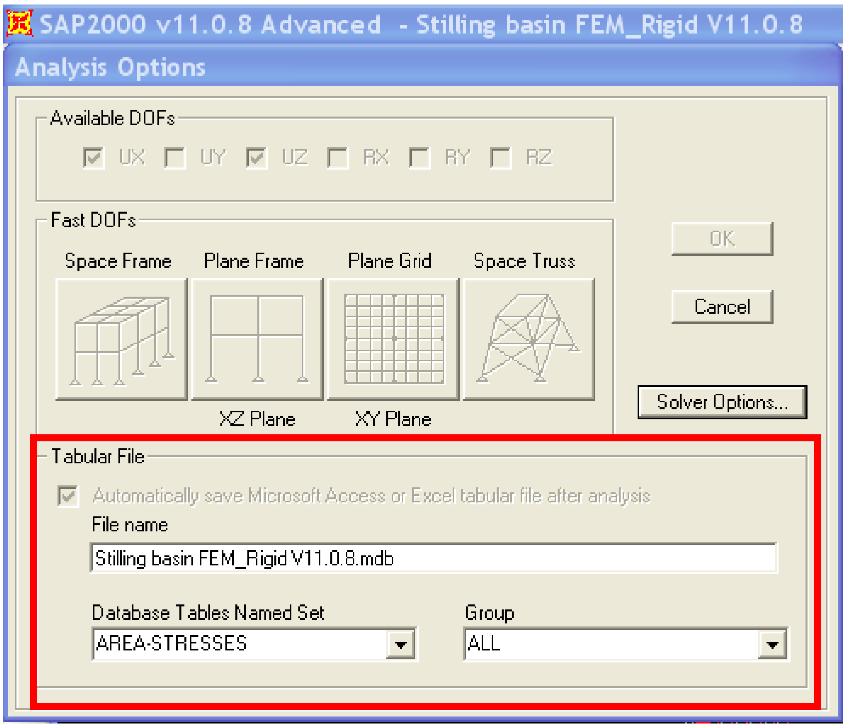


Figure 3.46: View to Specify output file

An Access or Excel file with the desired output will be created after the analysis has been completed.

Step N° 08: Starting Another Analysis

When the analysis is performed, Etabs "locks" the model . Thus, during the entire stage of visualization of the results, it is impossible to modify the model (geometry, connectivity, materials, supports, loads, etc ...). The lock icon, in the toolbar at the top of the screen, is pressed . To make changes to the model, it must be unlocked by clicking on this icon. In this case, all the analysis results will be destroyed. Once unlocked, the model can be edited and another analysis can be performed.

GENERAL CONCLUSION

GENERAL CONCLUSION

A building structure is a very complex industrial product, it is often composed of several dozen or even hundreds of elements (single, bi and three-dimensional), with dozens of nodes, tens of supports, distinct loads in nature (static and dynamic, current and rare, forces and couples...), in direction (vertical and horizontal), and in mode of application (pointed and distributed, uniform and non-uniform, etc). The task of designing and dimensioning these structures is often described as delicate, and demanding in terms of time and human effort, as it requires a lot of mental work, a large amount of knowledge (regulatory, normative, practices of construction companies, properties of materials and construction products, etc.), problems, physical analysis and mathematical calculations, to solve.

Given the difficulties encountered by the structural engineer, and given the complexity of the problem, the use of design assistance tools, and even drawing aid tools, is highly appreciated. These are the advantages of computing: quick analysis, comfortable work, precise results, etc.

This handout represents an effective tool that accompanies civil engineering students during their preparation of the bachelor's degree in civil engineering and in their professional environments. The design of concrete structures and their reinforcement is presented in a simple manner with didactic examples. It familiarizes the student with the modeling of simple structures and allows them to have an overview of more complex structures.

This handout serves as an overview to Etabs software, offering step-by-step instructions to help students develop their first model. Its purpose is to demonstrate the program's fundamentals and illustrate how easily and quickly models can be created. It emphasizes hands-on experience with Etabs as the most efficient way to become familiar with the program. While acknowledging Etabs' versatility and power, the present handout does not comprehensively document all its capabilities but provides a brief overview and commentary. It suggests using this research alongside other Etabs documentation for a complete understanding.

In brief, it can be said that this computer tool is very advantageous, as it ensures the speed (time savings), user comfort, as well as the accuracy and precision of the results provided. Also, the quality with a good presentation of these results in the form of tables, graphs, files, and on screen..., it is excellent. Thus, the efforts and time of the civil engineering structure designer will focus much more on intellectual issues such as finding optimal solutions and decision-making.

The author express hope that users will enjoy and benefit from this tutorial as a starting point in exploring the features of Etabs.

REFERENCES

REFERENCES

[1] E. L. Wilson and I. Computers and Structures, *Three Dimensional Dynamic Analysis of Structures: With Emphasis on Earthquake Engineering*. Computers and Structures Incorporated, 1997.

[2] NORMAN H. TAYLOR, “Review of Input and Output Equipment Used in Computing Systems Joint,” in *AIEE-IRE-ACM Computer Conference*, 1953.

[3] J. Allen and P. Kouppas, “Computer Aided Design: Past, Present, Future,” 2012, pp. 97–111.

[4] Douglas Taylor Ross and Massachusetts Institute of Technology, “Computer-Aided Design: A Statement of Objectives,” *Cambridge, MA M.I.T. Electron. Syst. Lab.*, p. 29, 1960.

[5] I. Computers and Structures, “Introductory Tutorial for SAP2000:Linear and Nonlinear Static and Dynamic Analysis and Design of Three-Dimensional Structures,” Berkeley, California, USA, 2011.

[6] B. S. Gan, “Finite Element Formulation of Beam Elements BT - An Isogeometric Approach to Beam Structures: Bridging the Classical to Modern Technique,” B. S. Gan, Ed., Cham: Springer International Publishing, 2018, pp. 61–126. doi: 10.1007/978-3-319-56493-7_3.

[7] I. Computers and Structures, “Linear and Nonlinear Static and Dynamic Analysis and Design of Three-Dimensional Structures: Introductory tutorial,” Berkeley, California, USA, 2004. [Online]. Available: www.csiberkeley.com

[8] K. D., R. Madihalli, M. S. L., N. K. B. M, and H. C. R., “Study on Influence of Rigid and Semi Rigid Diaphragm System on RC Structure,” in *2022 International Interdisciplinary Humanitarian Conference for Sustainability (IIHC)*, 2022, pp. 7–11. doi: 10.1109/IIHC55949.2022.10060573.

[9] H. R. Tanuwidjaja, G. K. Santoso, and E. Tanuwidjaja, “Rigidity Boundaries of Floor Reinforced Concrete Diaphragm,” in *Proceedings of the 6th International Conference on Civil, Offshore and Environmental Engineering (ICCOEE2020)*, B. S. Mohammed, N. Shafiq, S. Rahman M. Kutty, H. Mohamad, and A.-L. Balogun, Eds., Singapore: Springer Singapore, 2021, pp. 476–483.

[10] E. Saliklis, “Rigid Diaphragms BT - Structures: A Studio Approach,” E. Saliklis, Ed., Cham: Springer International Publishing, 2020, pp. 187–237. doi: 10.1007/978-3-030-33153-5_8.

[11] E. Oñate, *Structural Analysis with the Finite Element Method Linear Statics*. in Lecture Notes on Numerical Methods in Engineering and Sciences. Dordrecht: Springer Netherlands, 2013. doi: 10.1007/978-1-4020-8743-1.

[12] RPA99, “Règles Parasismiques Algériennes,” *D.T.R. –B.C. 2.48. Natl. Cent. Earthq. Appl. Res. (CGS), Algiers, Alger. (OPU). (ISBN 9961-923-13-8)*, 2003.

[13] Jean-Pierre Mougin, *Beton Arme. Bael 91 Modifie 99 Et Dtu Associes, 2eme Edition*,

Eyrolles. 2000.

[14] EC 8, *Eurocode 8: design of structures for earthquake resistance Part 1: general rules, seismic actions and rules for buildings*. European Norm. European Committee for Standardization, 2004.