

Starting Guide



Table of contents

1. INTRODUCTION	4
1.1 Welcome to Advance Design	4
1.2 About this Guide	4
1.3 Technical Support	5
2. WHAT IS ADVANCE DESIGN	6
3. INSTALLING ADVANCE DESIGN	7
3.1 System Requirements	7
3.2 Advance Design Installation	7
4. STARTING ADVANCE DESIGN	8
4.1 Project Management	8
5. Advance Design Environment	11
6. MODELING: CREATING THE DESCRIPTIVE MODEL	13
6.1 Advance Design Elements	13
6.1.1 Creating elements	14
6.1.2 Definition of element properties	16
6.1.3 Systems of elements	17
6.1.4 CAD functions	18
6.1.5 Generating loads	20
6.2 Defining Analyses	25
6.3 Model Verification	26
7. ANALYSIS: MESHING AND CALCULATION	27
7.1 Creating the Analysis Model	27
7.2 Meshing	29
7.3 Calculation	31
7.3.1. Finite elements calculation	31
7.3.2 Reinforced concrete calculation	33
7.3.3 Steel calculation	34
8. RESULTS POST-PROCESSING	
8.1 Graphical Visualization of Results	36
8.1.1 Result curves	39
8.1.2 Stresses diagrams	42
8.1.3 Post-processing animation	43
8.2 Design Post-processing	44
8.2.1 Reinforced concrete results	44
8.2.2 Steel results	47
8.2.3 Steel elements optimization	48

	8.2.4 Saved post-processing views	
	8.2.5 Reports	50
9	. DESIGN MODULES	
	9.1 Concrete Elements	
	9.2 Steel Connections	59
	9.3 Output: Calculation Results Tab	61
	9.4 Advance Design Modules Reports	
	9.5 Masonry Wall Module	
	9.5.1 Workflow	63
	9.5.2 Calculations and results	66

1. Introduction

1.1 Welcome to Advance Design

From modeling to structure calculation, result post-processing, and structure optimization, <u>Advance</u> <u>Design</u> offers a complete environment for the static and dynamic analysis of 2D and 3D structures using the finite element method.

This software also provides advanced design capabilities for steel, reinforced concrete, and timber structures. The verification of steel elements starts from an initial dimensioning and may continue with several successive optimizations. The reinforced concrete design determines, by several available methods, the theoretical reinforcement area, and the reinforcement ratios of concrete elements.

Advance Design is a new generation of analysis software, improved with powerful and innovative features:

1. Complete integration of finite elements / reinforced concrete/steelwork/timber analysis modules

2. Possibility to perform various tasks, such as:

- Model your structure with the assistance of various CAD tools (workplane, coordinate systems, snap modes, etc.) and CAD functions (extrude, subdivide, trim, extend, create symmetries, etc.).
- Input structure assumptions (materials, cross-sections, loads, analysis types).
- Create a mesh using two powerful mesh engines (Advanced and Standard Mesh).
- Calculate the structure using a new generation solver engine.
- View the results choosing from a large set of visualization options.
- Calculate and optimize reinforced concrete, steel, and timber structures according to standard regulations.
- Generate calculation reports using a variety of predefined result tables.

1.2 About this Guide

This guide describes the main functions and interface of Advance Design, and, through a few small examples, the program's working process. The examples follow each description of Advance Design functions.

This guide is a brief introduction to Advance Design and not all its features are described. For detailed information regarding the program functions, refer to the <u>Advance Design Help</u>.



1.3 Technical Support

GRAITEC technical support is available by phone, fax, or email. To reach GRAITEC technical support:

- **Ribbon**: **Manage** tab: click **Technical support** to send an email to GRAITEC.
- **Menu**: select **Help** > **Technical support**. A template email is sent to the technical support team who will quickly solve the problem and provide a precise answer. The model of the current project is automatically archived and attached to the message.

2. What is Advance Design

Advance Design, structural analysis software designed for the civil engineering field, offers a complete and fully integrated environment from the structure modeling to the result post-processing and structure optimization.

It provides a complete range of functions specialized in advanced CAD modeling, meshing, calculation, verification, and optimization of reinforced concrete, steel, and timber structures, result post-processing, and generating high-quality reports.

The study of a project in Advance Design environment is designed in four operating modes: **Model**, **Analysis**, **Design**, and **Document**.



Four operating modes: Model, Analysis, Design, and Document

3. Installing Advance Design

3.1 System Requirements

To successfully install Advance certain requirements must be met. For complete details, access the <u>Installation guide</u> or <u>www.graitec.com/advance-installation</u>.

3.2 Advance Design Installation

Before installing Advance Design:

- Make sure you have administrator rights under Windows.
- Close all open applications.

Proceed with the installation as follows:

1. Access <u>Graitec Advantage</u> on your browser and log in with your credentials. Find product releases sorted by type and year, under the **Downloads** section. You can either download a DVD ISO image of the software or download and run the online installer. If the AutoPlay tool on the computer is switched off and thus the setup does not start automatically, use the **Run** command:

- From the Windows menu, select: **Start** > **Run**.
- In the **Run** dialog box, click **Browse** to select the SetupAdvance.exe program. Click **<OK>**.

2. Select the installation language and click **Install products**.

3. On the next screen, select Advance Design and click **Next**.

4. Read the license agreement. Select I **accept** to agree to the specified terms and click **Next** to continue.

5. On the next screen, select the interface language and the installation path.

- To select the interface language, click **Customize**. In the next dialog box, select the interface language and the local settings for each installed application and click **<OK>**.
- To change the destination path, click . In the next dialog box, enter a path or select a different folder in which to install Advance and click **<OK>**.

6. Click **Install** to start the installation.

7. Wait a few moments while Advance Design is installed on the computer. Click **Exit** when the installation is complete.

After installing Advance Design, a license is required to use the software. The license is activated based on the activation code and the serial number provided by the dealer. Once the license is successfully activated, the software can be used according to the license rights.

Without the authorization code, a temporary license for 15 days may be installed. The activation process starts with launching Advance Design. Follow the procedure described in the Installation guide.

GRAITEC

4. Starting Advance Design

Advance Design can be launched using various methods:

- 1. From the Windows Start menu, select **Programs > Graitec > Advance Design**.
- 2. Double click the Advance Design icon on the desktop.
- 3. To start another work session simultaneously:
 - Double click an existing .fto file in its disk location.
 - Double click the Advance Design icon on the desktop.

4.1 Project Management

Each time Advance Design is launched, the Start-up Page helps you to create and configure new projects while managing and providing fast access to the existing ones.

The Advance Design start page has a left-side panel and two slides: CREATE and LEARN.

Example: Configuring a new project

1. On the left side panel of the CREATE page, click **Configuration**.

2. In the *Localization settings* dialog box make the following settings:

- Select the language to be used for the interface and the calculation reports.
- Select the Combinations standard: EC0 General.
- Select the Seismic standard: EC8 General.
- Select the Climatic standard: EC1 General.
- Select the Reinforced Concrete standard: EC2 General.
- Select the Steelwork standard: EC3 General.
- Select the Timber standard: EC5 General.
- Select the Masonry standard: EC6 General.

	Localization con	figuration	23
Standards	Configuration		
Axes convention		Europe 🔹	
	Languages		
	Canguages	English (INT)	
	Reports Language	English (INT)	
	Standards		
		NAD used	for Eurocodes
	Combinations	EC0	al 🕞
		NAD used	for Eurocodes
	Seismic	🖸 EC8 💿 Gener	al 🔹
		NAD used	for Eurocodes
		EC1	al 💽
		NAD used	for Eurocodes
	Reinforced Concrete	EC2	al 🔄
	Charlenade	NAD used	for Eurocodes
	Steelwork	EC3	al 🔄
	Teher	NAD used	for Eurocodes
	Timber	EC5	al 🔄
	— —	NAD used	for Eurocodes
	Masonry	EC6	al
L		OK Cance	I Help

Configuring a new project

3. Click **OK** and then select **New** from the left side panel to open a new project.

4. Enter details about the current project: name, lot, address, etc. These details are displayed in the reports generated for the current project.

Project Set	tings	×
Name	Advance Design Project	
Lot		
Address		-
City		
No 123	Phase I Variant A	
	List of business stakeholders	
All wizard p	arameters are accessible on the 'File \setminus Project settings' menu.	
Do not	display	
		_
	< Back Next > Cancel	

Project settings

Also, you can open the List of stakeholders dialog box where you can enter details about stakeholders and browse for an image file to add it as a project logo. When the Show in cover sheet option is enabled, the information provided in this dialog box is displayed on the report cover sheet.

5. Click **Next** to define the general structure configuration.

Project Settings			×
Workspace Mode: C Truss structure Bending rigid structure Default view Characteristics Reference temperature Default material Default cross section Units m, kN, kN*m, T, *	plane grid PERSPECTIVE View 10 °C C25/30 C25/30 Modify	© 3D	
		< B	ack Finish Cancel

Project Settings: Workspace, Characteristics, Units

5. Advance Design Environment

Advance Design offers a complete environment for modeling, analysis, design, and result post-processing - all fully integrated into the same interface.



Advance Design- General presentation

No.	Interface component	Main function / use
1	Quick Access Toolbar	Element integrated into the title bar and designed to provide one- click access to basic commands, such as "Save", "Redo" or "Print". You can customize it by adding the most used commands/features from the ribbon tabs.
2	Main Menu button	Clicking this top-left corner button displays a list of the Advance Design commands. The menus are listed from top to bottom considering the order of the work process steps. Position your cursor over a specific menu to further access specific commands.
3	Ribbon	A more user-friendly alternative to the previous standard menu bar, this interface component improves Advance Design usability by grouping commands into logical tabs and categories. Each tab in the new Advance Design ribbon allows access to several panels of commands. You can display or hide the tabs and their panels in the ribbon, by double-clicking on each tab.
4	Toolbars	Different types of commands are grouped in toolbars, which can be easily displayed and positioned (i.e., floating or docked) by drag-and-drop in the application environment. The toolbars that are active only in certain steps of the project (such as Modeling , Analysis Settings , Analysis - F.E. Results , etc.) are automatically displayed or hidden, to optimize the workspace during the work process.
5	Project Browser	The main control center of your Advance Design project, this tree- structure navigator displays different content in each working step of the project allowing quick access to commands varying from object selection to structure meshing or results post-processing.

GRAITEC

		The attributes of all the model entities can be viewed and modified
6	Properties window	in the properties window. The properties are displayed in a tree
		structure in various categories. The properties window is
		displayed dynamically when an element (or a drawing tool) is
		selected and provides access to the common properties of a
		selection of elements of the same type.
		The status bar displays information regarding the program status
7	Status bar	during different phases of the project. It also contains buttons that
		provide access to the configuration of certain parameters: snap
		modes, objects tooltip content, current coordinate system, and
		working units.
		The graphic area represents the design area of the application; it
8	Graphic area	provides easy and intuitive use of CAD commands and a realistic
		rendering of the model. It also allows you to perform practical
		actions like element drawing or selection.
		The graphic area can also be split into several viewports (from one
		to four); each of these viewports can have different display
		settings and a different viewpoint (i.e., zooms on a certain part of
		the structure, realistic or simplified rendering, etc.).
		The default workplane of the drawing area assists in structural
		modeling. The workplane's parameters can be defined and the
		workplane is easily hidden or displayed during the work process.
0		I he command line informs about the status of an action, assists in
9	Command-line	the drawing process, informs about errors, etc. It contains three
		tabs:
		- Information: displays the status of the current operations.
		- Errors: displays warnings and error messages.
		- Edit: this allows the dialog between user and application;
		provides the option to draw/modify objects by typing
		parameters in the dialog area of the command line.
		The global coordinate system is represented by a three axes
10	Coordinate system	symbol permanently displayed in the graphic area. It is also
		possible to create one or several user-defined coordinate systems
		(Cartesian or polar).

The program interface is intuitive and enhanced for easier manipulation of its different components and commands (i.e., advanced docking, undocking, auto-hiding, tabbing, etc.).

6. Modeling: Creating the Descriptive Model

The structure modeling may be done entirely with the use of various CAD tools via the graphic area, where a 2D or 3D representation of the model can be visualized at any time.

The various zoom and view commands (e.g., rotate around the model, predefined views, etc.) provide fast and easy manipulation of the graphical elements.

6.1 Advance Design Elements

Advance Design provides a complete library of structure elements, supports, and geometric entities.



Example	Element type
	Linear elements (bar, beam, short beam, variable beam, tie, strut, cable)
	Planar elements (membrane, plate, shell, plane strain, steel deck)
^{~~} ~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~	Supports (point, linear and planar supports, which can be rigid, elastic, traction only / compression only, or non-linear)
×	Load areas: elements used for the distribution of loads on the supporting elements
* *	Points
\square	Lines and polylines
	Grids
22 ^m +1.5 m	Dimension lines

GRAITEC

> 6.1.1 Creating elements

Elements are drawn in the graphic area using the keyboard (by typing coordinates on the command line) or the mouse, relative to the workplane's points or existing entities. Advance Design also provides various automatic drawing tools (e.g., generate elements on selection, portal frames, and vaults generators, etc.).

Example: Creating structure elements

1. On the Home tab, Linear & Planar panel, select Linear element.

2. In the graphic area (XZ plane), click to define a column at **00** coordinates for the first extremity and **07** for the second one.



3. With the linear element drawing tool still active, type on the command line the coordinates for the second column: **12 0** for the first extremity and **12 7** for the second one. Input a space between coordinates and press **Enter** after each extremity definition.



4. Draw the upper beam between the two columns, using the "Extremity" snap mode:



5. Draw a 3 meter high column placed at **60** XZ coordinates.



6. Create a storey beam: with the linear element drawing tool still active, press **<Alt + S>** to access the snap modes dialog box; select the perpendicular snap and draw the beam as shown below.



7. On the **Home** tab, **Supports** panel, select **Rigid Point**; in the drawing area, click the bottom extremity of each column, to create supports.



> 6.1.2 Definition of element properties

The attributes of each element are defined in the properties window (e.g., name, ID, and different parameters). By default, the properties window appears when an element is selected and auto-hides when it is empty.

Example: Defining the element properties

1. Click the two columns and the beam of the main portal frame to select them.



2. In the properties window make the following settings:

- Material: C25/30.
- Cross-section: **R60*90**.

3. Select the storey column, and, in its properties window, select the **S235** material. To define the element cross-section, click to access the section library and select **IPE200** from the "European Profiles".



4. Proceed in the same way to define the material and cross-section for the storey beam: **S235 - HEA200**.



> 6.1.3 Systems of elements

The structure elements that are created (also geometric elements, help entities, etc.) are stored in the Project Browser, in the Model mode. The Project Browser's context menus for each of its items provide rapid access to different modeling commands and advanced management of the elements (i.e., hide/display, select, delete, the group in systems, etc.).

The system concept helps define the behavior of different groups of elements (e.g., assign design templates). Different operations are easily performed on a group of elements using their system's context menu commands. The level function available in the system's properties window defines the level settings and thus creates faster and easier structure elements on different altitudes (levels): a column using one click, a wall using two clicks.

Example: Creating a system of elements

1. In the Project Browser, right-click **Structure** and select **Systems management / Create a subsystem** from the context menu.

2. Type the system name: **Portal frame**.

3. Select the **Portal frame** system, and using the steps described above, create the following subsystems: **Columns, Beams, Supports, and Load Areas**.

4. Create a **Storey** system under **Structure** with the **Columns, Beams, Slabs, and Supports** subsystems.



Creating a system of elements

5. In the Project Browser, select the two columns of the portal frame and drag and drop them to the **Portal frame > Columns** system.



Using the same method, place all the elements of the model in the corresponding systems.

6.1.4 CAD functions

The graphical input of the model is very easy and accurate, using advanced CAD commands. Easily copy (i.e., by rotation, translation, or symmetry), move, extrude, trim, or extend, subdivide, cut, create openings, etc. using a large set of commands accessible from various locations (i.e., menus, context menus, toolbars, etc.).

Home	Objects	Utilities Anal	ysis Results	Documents	BIM Manage	Design	n Modules									
Application	+ /	A B			8Q *	10	🔂 🕂 -	11-	🔄 Select by Criteria	🚰 fi ^r	🚸 - 🔲	⊕	/ 🕫 🏟	n.		Text Size
C Display	₩ 🖊		Diaid Diaid	Dinid Load	U Combinations			•∕•	👋 Invert Selection	ារ សា	👩 - 💽	±	2 🗭	1 -	Norifu	
📤 Structure *	14 📀	Element + Element +	Point + Linear + P	Manar Area	loa	ad -	Lopy A -	₽	Select by 🔹	ritter 🧔	view *	12	P		verny	
Project Settings	Lines	Linear & Planar	Supports	Load Area	Load Case & Load	s	CAD Function	ns	Selection	Isolate / Display	Views	Plan	Render Modes	Snap		Slider



Example: Copying elements

First, define a 3D view of the workplane: on the **Home** tab, **Views** panel, click (or press **<Alt + 6>**).

1. Press **<Ctrl + A>** to select all the elements of the model.

2. Right-click the drawing area and select Copy (or press < Insert>).

3. In the 'Multiple copy' dialog box, define the copy parameters:

- Copy by Translation.
- Vector: **060**.
- Number: **3**.

V Translation	
Mode:	•+• • •-•+ ↓ ↓ ↓ •
Vector:	0, 0, 0
Distances:	
& Rotation	
Mode:	· · · · · · · · · · · · · · · · · · ·
Origin:	0, 0, 0
Axis:	0, 0, 1
Angle:	0 °

Copy function

- 4. Click **Preview** to display the result.
- 5. Click **Copy** to apply.



Create the rest of the storey elements:

- In the Project Browser, select the **Storey > Beams** subsystem.
- Select the linear elements drawing tool and draw two longitudinal beams (S235 with HEA200 cross-section).
- Select the first transverse beam and make 2 copies in the **0 2 0** direction.
- Select the two copied beams and define, in their properties window, an **S235** material and an **IPE200** cross-section.
- Make 2 copies of the selected beams in the **0 6 0** direction.

GRAITEC



> 6.1.5 Generating loads

Loads are generated and organized using the **Project Browser**. Loads are grouped in the Project Browser under "*Loading*" in load cases (i.e., self-weight, static, seismic, etc.) and case families (permanent loads, live loads, snow, wind, temperatures, etc.). Each case family may contain several load cases, and each load case may contain several loads.



Loads are grouped in the Project Browser under "Loading" in load cases

Loads are generated via the graphical input, using the load creation tools available on the **Home** tab -**Load Case & Loads** panel, in the Project Browser or menus. The automatic tools (i.e., pressure loads generator, climatic loads generator, loads on selection, etc.) can also be used.

The parameters of loads, load cases, and case families are defined in their properties windows. The loads are managed using their context menu commands in the Project Browser.

Once loads have been defined, load combinations and envelopes can be created (using the Project Browser or the **Analyze** menu commands).

Using the combinations manager, combinations are manually created and the standard combinations available in Advance Design are loaded.

GRAITEC



Generate a new Case Family

Example: Generating loads

• Generate the self-weight

From the menu, select **Generate > Load > Dead Loads**. The 'Dead loads' family and a dead load case are automatically created in the Project Browser.

• Create a live load

First, create the storey slab. In the Project Browser: select the **Storey > Slabs** subsystem. On the **Home** tab, **Linear & Planar** panel, select **Planar element** and draw the slab as shown.





In the slab's properties window define:

- Material: C25/30.
- Thickness: 15 cm.
- Design > Concrete Design > Cracking > Reinforcement definition > in and make the following settings:

G	lob	al definition				
		Direction	Туре	Diameter	Dist	
		Along x - sup	Bars HB	Ø 8	5.00 cm	
		Along x - inf	Bars HB	Ø 8	5.00 cm	
		Along y - sup	Bars HB	Ø 8	5.00 cm	
	•	Along y - inf	Bars HB	Ø 8	5.00 cm	



1. In the Project Browser: right-click Loading and select Create a case family from the context menu.

2. In the displayed window, select Live Loads.

3. Click **<OK>**. A live loads family and a corresponding case are created in the Project Browser.

4. Select the **2L** live load case in the Project Browser.

5. In the drawing area, select the storey slab.

6. Right-click and select Loads/selection from the context menu.

7. The planar load's properties window is automatically displayed: input the loads' intensity on **FZ: - 5 kN/m**.

8. Click **<OK>**.



• Generating snow loads

First, create two beams and a load area on the portal frame:

1. In the **Portal frame > Beams** subsystem create two longitudinal portal frame beams, using a **C25/30** material and an **R40*60** cross-section.



2. In the Project Browser: select the **Portal frame > Load Areas** subsystem. Select the two longitudinal beams; right-click and select **Load Areas / Selection** from the context menu. On the **Home** tab, the **Render modes** panel, enable the 'Axes' rendering mode. This provides a view of the span direction on load areas. In the load area properties window: set the span direction towards the longitudinal beams, considering the load area's local axes.



Loads Distribution

3. In the Project Browser: right-click **Loading** and generate a snow family using the same steps described above. A snow family and a snow load case are created in the Project Browser.

4. In the snow family properties window:

• Enter **0.52** kN/m² in the 'Snow load' field.



5. To automatically generate the snow loads on the load area, select **Generate > Load > Climatic loads** from the menu.



Loads presentation

Example: Creating load combinations

- 1. In the Project Browser, right-click **Combinations** and select **Properties** from the context menu.
- 2. In the 'Combinations' dialog box, click **Simplified Comb**.
- 3. Define the Live loads and Snow loads as predominant actions.
- 4. Click Generate.

	Combinations options
Project Situation	n
ULS:	EQU STR/GEO Equation 6.10
SLS:	Characteristic X Frequent Quasi-permanent
	Dead Loads y, y,
edominant action	Unfavorable • 1.1 • 1.35 •
	Live loads
V	A Category: housing, residential areas V 0.7 0.5 0.3 1.5 V 1.5 V
	Snow loads
V	Other CEN member states in places locate 0.5 0.2 0 1.5 1.5 1.5
	Wind Loads V_0 V_1 V_2 Y_E Y_S
×	Wind loads on structure Image: Constructure Image: Constructure </td
	Temperature Loads \V_0 V_1 V_2 Y_E Y_3
×	Temperature (besides fire) inside the build 0.6 0.5 0 1.5 1.5 1.5
	🗙 Cancel 🐌 Reset 🗸 Save 🖌 Generat

Creating load combinations

6.2 Defining Analyses

During the modeling step, Advance Design provides commands for defining several types of analyses (i.e., modal, buckling, static nonlinear) and for the concrete and steel design settings.

Access the **Settings** menu to select the desired analysis. For each type of analysis, a default analysis case is also automatically created.



Create a modal analysis

Manage analyses using the Project Browser's commands. View and select the created analyses with the project browser.



The analyses cases parameters are defined in the properties window.

Example: Defining a modal analysis

1. Menu: select **Settings > Modal Analysis**.

2. In the Project Browser, a modal analysis family and the 'Modes' case are placed in the 'Settings' group. Select the 'Modes' case to display its properties window.

3. Define the modes' parameters:

- The number of vibration modes: **10**.
- Masses definition: select masses obtained by combining static loads from the drop-down list and, in the 'Combinations' field below, click is to access the masses combinations dialog box. Define here the following combination: 1*1D + 0.6*2L.

 Envelopes Combinations (Number: 12) Modal Analysis Modes Construction Stages Saved views 	Modes Target Number Participation ratio Masses Definition Combinations	Number of modes 10 95 % masses obtained by combinin 1.00°1D + 0.60°2L
Combinations Selection definition Available analyses 1 2 3 Snw	Selected analyses 1 D : 1 2 L : 0.6 Remove <	×
	ОК	Cancel

Defining a modal analysis

• Mass percentage on Z: 0%.

6.3 Model Verification

At any time during the modeling step, the model's coherence and integrity can be verified with the

verification function. Access the **Analyze** menu > **Verify** command or click (Verify) on the **Home** tab. If there are errors or warnings, they are displayed on the command line. If there are no errors, a confirmation message will be displayed.



7. Analysis: Meshing and Calculation

In the next step, after the verification of the model's coherence and validity, the program creates the analysis model. The structure meshing and the model calculation are performed using the defined analyses (i.e., finite elements calculation and concrete/ steel verification).

7.1 Creating the Analysis Model

To mesh and calculate the structure, it is necessary to create the analysis model. Once the model's validity is verified, access the **Analyze > Create the analysis model** command, or, in the Project

Browser, click

A wizard provides options to combine the desired operations (i.e., verify, mesh, finite elements calculation, reinforced concrete calculation, etc.) into a calculation sequence that is automatically performed.



Creating the Analysis Model

The analysis model's components are controlled and viewed in the Analysis mode in the project browser. The available context menu commands for each element of the project browser manage the analysis operations.



Analysis functions

After the analysis model creation, new panels and commands are available (e.g., the **Analysis Model** panel from the **Analysis tab**), while the modeling tools are inactive.



Analysis Model- Mesh options

7.2 Meshing

Two different mesh engines are available in Advance Design: 'Grid' and 'Delaunay'.

The finite element meshing is performed using the global mesh settings (defined via the **Options > Mesh** command) and the mesh parameters defined for each element (available in the properties window). The mesh parameters of each element are defined using the simplified method (i.e., a meshing density along each of the local axes) or the detailed method (i.e., a meshing density for each of the element's sides).

Example: Defining the model meshing

• Create the analysis model and the model meshing

1. In the project browser, click 🔲 to access the 'Calculation sequence' dialog box.

2. Select Mesh and click <OK>.

Advance Design creates the analysis model and automatically performs the model meshing.

	Calculation Sequence	ce	23
Actions			_
Open the current	analysis model		
O Create a new ana	lysis model and launch the following	actions:	
	📝 Verify	RC calculation	
	🔽 Mesh	Steel calculation	
555	Evaluate	Timber Design	
EFE	FE calculation		
	Pushover Calculation		
	Run calculations for the select	ted construction stage	
	•		
		의 	
	Update the post-processing vi	ews	
			4
		OK Cancel	

Create the analysis model and the model meshing



• Modify the mesh density

1. From the menu, select Options > Mesh.

2. In the 'Mesh options' dialog box, modify the mesh density: in the 'Element size by default' field, input **0.5** meters.

3. Click **<OK>**.

Mesh				
Mesh type	De	launay	•	
Element type				
Triangles and Question	uadran	gles (T3-Q4)		
C Only quadrangle	s			
C Only triangles				
 Include loads in t 	he mes	sh (linear elements)		
 Include loads in t Include loads in t 	he mes	sh (linear elements) sh (planar elements)		
 Include loads in t Include loads in t Element size by defaul 	he mes he mes	sh (linear elements) sh (planar elements) 0.5 m]	
 Include loads in the second sec	he mes he mes t	sh (linear elements) sh (planar elements) 0.5 m 0.001 m]	
 Include loads in the second sec	he mes he mes	sh (linear elements) sh (planar elements) 0.5 m 0.001 m Define by default]	

Mesh options

• Recreate the mesh

On the Analysis tab, Analysis Model panel, select Mesh.

The meshing is modified according to the global settings.

7.3 Calculation

After meshing, Advance Design is ready to calculate the model. The 'Calculate' command provides access to the 'Calculation sequence' dialog box, where the calculations to perform is selected.

Calculat	ion Sequence	23
The mesh was already created	. The following actions are available:	
🖂 Mesh	Reinforced Concrete calculation	
Evaluate	Steel calculation	
☑ Finite elements calculation	Timber calculation	
Pushover Calculation		_
Run calculations for the selected cons	truction stage	
Update the post-processing views		
Update active calculation report		
	OK Cancel	

After meshing, Advance Design can calculate the model

> 7.3.1. Finite elements calculation

A powerful finite elements engine performs the model calculation according to the structure settings:

- Defined analyses (static and dynamic calculation, linear and non-linear analyses, large displacements, generalized buckling, etc.).
- Finite elements parameters of structure elements (defined in the properties window).

Before the calculation, it is possible to select the elements to calculate and the type of results to obtain, for optimizing the calculation speed and the memory usage.

G Home	Options - Application	23
Application	General Memory Results Graitec BIM Project Browser Melody	
Display Structure * Project Settings Project Settings Project Browser ADVANCE I Model Analysis Model Analysis Model Analysis Model Cading Codading Codading Codading Codading Codading Construction Settings Construction Saved views	Save results Families of elements Ø Node Ø Support Ø Linear Bement Ø smoothed Ø smoothed Ø smoothed I use T6Q9 meshing in calculation Consider reactions on supports Smoothing of extreme values Smoothing of extreme values Smoothing of extreme values Smoothing of extreme values Smoothing of interne values	
	OK Canc	el

Application - general settings

During the analysis step, it is possible to specify the analyses to calculate (using the Project Browser commands).



Advance Design can group analyses in calculation phases and calculate them step by step (allowing property modifications for each phase).

Home Objects Utilities Analysis Results	Documents BIM Manage Design Modules		
Analysis Nodel View V DOF DOF Elastic Create a Restraint Constraint Link link at node Objects	Verify Mesh Mesh Calculation Analysis Model	Settings Calculation RC Design	on Settings Calculation Timber Design
USER view	Calculation Sequence	? ×	
	Phases Phase configuration		
	Phase No.1 Cases/combinations to Phase No.3 1.0 Phase No.3 1.1 Phase No.3 1.1 Intervention 1.1	Phase in progress 3::5 3::5 102 - 1.3x(1 D)+1.5x(2 L)]	

Analysis- Calculation by Phases

> 7.3.2 Reinforced concrete calculation

The reinforced concrete engine calculates the reinforcement of concrete elements by serviceability limit states (SLS) and ultimate limit states (ULS and AULS) and verifies the concrete cross-sections using interaction curves.

The reinforced concrete calculation is performed only if the regulatory load combinations are created, and the calculation of the finite element was run. The reinforced concrete calculation considers the global and local concrete design settings:



➢ 7.3.3 Steel calculation

Advance Design provides a steel calculation engine, which performs the calculation of steel elements according to standard regulations. The steel expert verifies deflections, the cross-section's resistance, and the element's stability according to second-order effects (buckling and lateral-torsional buckling) and optimizes the steel shapes.

The steel calculation is performed only if the standard load combinations are created, and the calculation of the finite element was run.

The steel calculation considers the global and local steel design settings:



Note: After the calculation, the results can be viewed and the element parameters modified, if necessary. The desired calculations can be iterated until the appropriate results are obtained.

Example: Running a complete calculation sequence

1. From the **Analyze** menu, select **Calculate**.

2. In the 'Calculation sequence' dialog box, select:

- Finite elements calculation,
- Reinforced Concrete calculation,
- Steel calculation.
- 3. Click **<OK>** to launch the selected operations.

Calculation Sequence						
The mesh was already created. The following actions are available:						
Mesh	Reinforced Concrete calculation					
Evaluate	Steel calculation					
Finite elements calculation	Timber calculation					
Pushover Calculation						
Run calculations for the selected cons	struction stage					
Update the post-processing views						
Update active calculation report						
	OK Cancel]				

Calculation Sequence

The command line displays the performed operations and a message when the calculations are done.

Command Line	д	×
Launch reinforced concrete calculation Reinforced concrete calculation finished		^
Steel calculation finished		
		v

Command Line presentation

8. Results Post-processing

The phase following the model calculation, also called the post-processing step, displays the results on the graphical model or as calculation reports, results from curves on the selected elements, etc.

8.1 Graphical Visualization of Results

A new set of tools and commands are active during the results post-processing step. They provide different modes to display the desired results. Several results visualization commands are available:

• On the panels in the **Results** tab, which appear automatically once the corresponding calculation is done.

	Results Do	cumen	ts	BIM	Mana	ige Des	ign Modules						
÷	Displacements	*		None	Ŧ	2		Reinforcement	Ŧ	2	Ŧ	14	— -
	5 NL :1 NL	Ŧ	0	D	Ŧ		0	None	Ŧ	14		l <u>∼</u> ×	
	Last step	*		D	*	Post Processing		None	F	Post Processing	Reinf.	Curves *	ColorMap Configuration *
FEM Results						RC Design Re	sul	ts		Pos	t Processing		

Results tab: FEM, RC Design, Steel Design Results

• From the element's context menu: it is possible to display in the graphic area the results of the selection. When a selection is not defined, the results are displayed on the entire structure.

	Cancel selection
	Zoom / Selection
	Cross Section Stresses
₩	Result Curves
	Create DOF restraints on selection
	Create DOF constraints on selection
	Connections •
G	Export to Design Modules
ጰ	Open with Design Modules
P	Export to ACB
	Display mesh on selection
	Selection •

 Using the results configuration dialog box, that provides a detailed configuration of the results display. Different visualization modes are available: colors, values, deformed shape, isovalues, iso-regions, vectors, etc.

37

GRAITEC

Results	Results	23
	Reinforcement Image: Section Cut Display mode Planar Bement Axb Iso regions Section Cut Axt Iso map Max (Axb, Ayb) Axt Max(Axb, Ayb) Max(Axb, Ayb) Max(Axb, Ayt) Values to the cent Values to the cent Values on grid - Mi Values on grid - Mi Values on grid - Mi Values on grid - Mi Values on grid - Ait Diagram scale 2 00	r E A
	OK Cancel	Help .::

Results Settings dialog

Example: Creating graphical post-processing of FE results

• View the displacement results on the storey slab

1. First, right-click in the graphic area and disable **Display nodes** from the context menu. Define a top view of the workplane: press **<Alt + 3>**. On the **FEM Results** panel, select the **Displacements** result type, the **D** planar elements results, and the combination no. **101**. Click **Post Processing** to perform the post-processing.

2. Define a (-1, -1, 1) view of the workplane: on the Home tab, Views panel, click \square . In the Result Settings dialog box - Options tab: select Display results on the deformed and Automatic scale of the deformed.

lysis	;	Results	Docume	nts	BIM	Manag	e Desig
n *	+ @	Displace 103 1.3x[ments 1 D]+1.5x[; ;	• 🛆 • 🥒 • 🖓	None None D	T T T	Post
FF An Pla Lo	RONT alysi anar I cal A	view s: 103 (1 Element xes	1.3x[1 D]+1 : D	l.5x[2	L]+0.75x	[3 S])	<u>L</u>
	-						

Post Processing result - Displacement on planar elements

3. Right-click the graphic area and deselect **Display the mesh** from the context menu.

4. Open the results configuration dialog box (press **<Alt + Z>**), access the **Options** tab, and select **Extreme values**.



View forces results on the concrete linear elements

1. Select the concrete elements using the selection by criteria: press **<Alt + S>**; in the 'Elements selection' dialog box, access the **Materials** tab and select **C25/30**. Click **<OK>** to apply.

2. On the **FEM Results** panel, select the **Forces** result type, the **Fy** linear elements results, and the combination no. **101**. Click **Post Processing** to perform the post-processing.



View forces results on the concrete linear elements

• View stresses on steel elements

First, right-click the drawing area and select **Cancel selection** from the context menu.

1. Define a new selection by criteria: press **<Alt + S>**; in the 'Elements selection' dialog box, access the **Materials** tab, and select **S235**. Click **<OK>** to apply.

2. On the **FEM Results** panel, select the **Stresses** result type, the **Sxx** linear elements results, and the combination no. **101**. Click **Post Processing** to perform the post-processing.



View stresses on steel elements

To clear the results displayed on the screen: hold down the **<Esc>** key for a few seconds.

> 8.1.1 Result curves

Different results (i.e., FE results as displacements, forces, stresses, and concrete reinforcement area) can be viewed using the "Result curves" command available in the post-processing step of the project.

Result curves can be obtained on linear and planar elements using section cuts. The result curves diagram is configured using various options available from the diagram's dialog box. The diagram can be saved as an image or it can be printed using specific commands.

Example: Displaying result curves on a section cut

First, create a section cut:

• Right-click the drawing area and select **Generate an entity > Create a Section Cut** from the context menu. Draw the section cut on the length of the storey slab as shown in the figure.



Positioning Section Cut

Then, select the analyses displayed on the curve and view the results:

- Press **<Alt + Q>** to access the 'Analyses and combinations' dialog box.
- On the **Forces**, tab click **None** to unselect all loads combinations, and then select only the **101** loads combination.

Analyses and Combinations (Alt+Q) ?										
Style Name Default New Rename Delete										
٩		Displacem	ents (15) Forces (1) Stresses (15)				Þ			
		ld.	Analysis/Combination Name	Details	Step					
		1	D	ECG						
		2	L	ECQ						
		3	Snw	ECN						
		0	Modes	Mode 1 : T = 0.232 s						
•	\checkmark	101	1.35x[1 D]	ECELUSTR						
		102	1.35x[1 D]+1.5x[2 L]	ECELUSTR						
		103	1.35x[1 D]+1.5x[2 L]+0.75x[3 Snw]	ECELUSTR						
		104	1.35x[1 D]+1.5x[3 Snw]	ECELUSTR						
		105	1.35x[1 D]+1.5x[3 Snw]+1.05x[2 L]	ECELUSTR			-			
	All		None Analysis type	 Codes or id 	dentifiers		•			
					ОК	Cancel				

Analyses and Combinations

- Select the section cut.
- Click in the **Results** tab, **Post-Processing** panel. The default result curves (**Mxx** and **Myy**) for the selected analyses are automatically displayed.



Results Curves on Section Cut 1

> 8.1.2 Stresses diagrams

The analysis of the stresses distribution on a given section is performed using the section stresses command. A stresses diagram is obtained. The diagram dynamically displays the stresses results on each point of the linear element.

Example: Displaying a stresses diagram

Select the storey beam on which to display the section stresses results:

- Position the mouse cursor above the desired beam; the tooltip displays the details of the element focused by the cursor.
- Press the <Tab> key to snap to different elements placed on the cursor trajectory; when the cursor focuses on the beam of interest (i.e., S235 material, HEA200 crosssection) - click to select it.



From the menu, select **Analyze > Cross Section Stresses**.

The sections stresses diagram is displayed in a new window. Use the slider to view the stresses on each point of the beam's length.



Linear elements stresses

> 8.1.3 Post-processing animation

An animation can be created in Advance Design, starting from the graphical results post-processing, following the results distribution and the deformed shape of the structure.

The **Animation** panel in the **Utilities** tab provides access to all necessary commands for creating and recording animations.



Animation functions

Example: Creating a post-processing animation

1. On the **FEM Results** panel, define the following results post-processing:

- Select the Eigenmodes result type.
- Select Eigen mode 3 from the analyses drop-down list.
- Click Post-processing.

2. Define a front view of the workplane: on the **Home** tab, **Views** panel, click ${}^{ imes}$.



Displaying results for modal analysis

3. On the **Results** tab, **Post Processing** panel, open the **ColorMap Configuration** drop-down list and select **Animation** to view the post-processing results in animation.

To stop the animation: press the **<Esc>** key.

8.2 Design Post-processing

In the post-processing phase, once the corresponding calculations are performed, the results of the concrete/steel verifications can be viewed. Moreover, the steel members of the structure can be optimized using the functions provided by these design modules.

For this purpose, a set of specialized panels and commands are available, fully integrated within the same interface.



> 8.2.1 Reinforced concrete results

The reinforcement results on concrete elements (i.e., reinforcement area, buckling lengths, reinforcement ratios) are viewed using the **Results** tab, **RC Design Results** panel, which is available when the concrete calculation is completed.



In the properties window of certain concrete elements (columns), the interaction curves issued from the reinforcement parameters, which are either determined automatically by the concrete module or set by the user, can be viewed. Therefore, it is possible to adjust, for example, a highly slender column exposed to oblique bending.

Example: Viewing the longitudinal reinforcement on beams

1. Define a (-1, -1, 1) view of the workplane by pressing **<Alt + 6>**.

2. In the Project Browser, select Portal frame > Beams system and press <Space>.

- 3. On the **Results** tab, **RC Design Results** panel:
 - Select the result type: Reinforcement.
 - Select the result on linear elements: Az.
 - Click **Post-processing** to perform the post-processing.

The longitudinal reinforcement is automatically displayed as diagrams. The result values appear in the color legend displayed in the graphic area.



Viewing the longitudinal reinforcement on beams

Example: Viewing the reinforcement results on a column

1. Select one column of the portal frame.

2. In the element properties window, go to the **Design** category > **Reinforcement** field.



Properties - Reinforcement options

3. Click for the 'Modification of longitudinal reinforcement' dialog box. The values of the real reinforcement and the calculated reinforcement for the selected column can be viewed.

Main reinforcement Secondary reinforcement Dameter 0 14 Sup. angle Straight bar Hf. level = 0.00 m Hf. level = 0.00 m Hf. level = 0.00 m Hf. level = 0.00 m Hf. level = 0.00 m Hf. level = 0.00 m Hf. level = 0.00 m Hf. level = 0.00 m Hating is 2 Along is 2 Calculated values Coros sectorn tool Interaction curve Interaction curve Interaction curve	4 Longi	itudinal reinfor	cement T	ransverse bar	\$
Dameter 0 14 Dameter 0 14 Sup. angle Straight bar Sup. angle Straight bar If angle <liif angle<="" th=""><th>Main reinforce</th><th>enert</th><th>-</th><th>- Secondary r</th><th>einforcement</th></liif>	Main reinforce	enert	-	- Secondary r	einforcement
Sup. angle Straight bar If. angle Straight bar Sup. level = 6.56 m Sup. level = 6.56 m If. level = 0.00 m	Diameter Ø 14 💌		Diameter	0 12 •	
If. angle Straight bar If. angle Straight bar Sup. level = 6.56 m Sup. level = 6.56 m Sup. level = 6.56 m If. angle Sup. level = 6.56 m If. Angle Nb. Ø Pp Nb. Ø Sec 4.014 + 6.012 If. level = 0.00 m Place Nb. Ø Pp Nb. Ø Sec 4.014 + 6.012 If. level = 0.00 m Along it 2 2 1 Theoretical R = 1.203.62 em Calculated values Coros sector tools Interaction curve OOOO If. If. angle If. angle	Sup. angle	Straight bar		Sup. angle	Straight bar
Sup. Invel = 6.36 m Sup. Invel = 6.36 m Inf. Newl = 0.00 m Inf. Newl = 0.00 m Place Nb. 0 Pp Along a: 2 Along b 2 Calculated values Cross sector tools Interaction curve Interaction curve	Inf. angle	Straight bar	•	Inf. angle	Straight bar
Hr. Jevel - 0.00 m Hr. Jevel - 0.00 m Place Nb. 8 Pp Nb. e Sec Along a 2 2 Along b 2 1 Calculated values Cross sector tools Interaction curve	5	o level - 6.56	-	5	up level = 6.96m
Place Nb. Ø Pp Nb. ø Sec 4 014 - 6 o12 Akong a 2 2 Theoretical R = 1283 520 mm Akong b 2 1 Read R+ 1294 34 mm²(100.83 Calculated values Cross sectorn tools Interaction curve 00006 II III IIII		f Xevel - 0.00 r	n ;	+	f level = 0.00m
Along a 2 2 Along b 2 1 Theoretical R = 1283 52 mm Casc 105) Real R = 1256 34 mm ² (100.83 Calculated values Cross sector tools Interaction curve 0000 II I	Place	Nb. Ø Pp	Nb. ø Sec		4 814 - 6 012
Along ib 2 1 Cate 1051 Pearl R+ 1254 34 mm ⁻¹ (100.83 Calculated values Cross sector fools Interaction curve 000000 01 03 0 00000 03 0 0 0 0 0 0 0 0	Along a	2	2	Theor	etical R = 1203.62 mm ^s
Calculated values Cross sector tools I I I I I I I I I I I I I I I I I I I	Along b	2	1	_	(Case 105)
Calculated values Cross sector tools Interaction curve				Real R+	1294.34 mm ² (100.83 %)
	Calculat	ed values	Gross sec	tion tools	Interaction curves
			90.00	-	

Detailed rebar definition on a column

5. Click Interaction curves to access the interaction curves window.



Interaction curves for column

It is possible to view the position of the force component relative to the interaction area. For advanced visualization options, double-click the diagram, and the interaction curve is displayed in a new window.

> 8.2.2 Steel results

During the post-processing step, after the steel calculation, the steel expert module performs the verification of deflections, section resistance, and element stability according to second-order effects (buckling and lateral-torsional buckling), and the optimization of steel shapes.

The steel results post-processing commands are available on the Results tab, Steel Design Results panel once the steel calculation is done.



The calculated buckling and lateral-torsional buckling parameters for each steel element can be viewed in the properties window.

Example: Verifying the steel elements' stability

1. On the Steel Design Results panel:

- Select the result type: **Stability.**
- Select the result: **Work ratio**.
- Click **Post-processing**.



Example: Verifying the steel elements' stability



- 2. Access the 'Results' dialog box by pressing **<Alt + Z>**.
- 3. Access the **Options** tab and select **Extreme values**.
- 4. Click **<OK>**.

> 8.2.3 Steel elements optimization

The steel design module verifies the steel elements according to parameters specified by the global steel settings.

The program identifies the steel shapes with a higher / lower work ratio than specified and suggests more appropriate cross-sections.

Calculation Settings		×
Verification Safety factors Optimisation Sort profiles Buckling Angle verification Calculation Sequence	Optimisation Optimisation mode Image: by element Image: by design template Image: per system Image: by sections Image: by name Optimisation criteria Image: per system Image: per system Image: Optimisation criteria Image: per system Ima	
	OK Cancel	Help

Steel- General settings

The suggested shapes can be accepted in whole or in part. Then, it is necessary to rerun the FE calculation and the structure optimization. These operations can be iterated until the appropriate work ratio is obtained for all steel shapes.

Example: Optimization of steel shapes

1. On the **Results** tab, **Steel Design Results** panel, click *I* . The 'Suggested shapes' dialog box appears. The steel sections with a work ratio out of the specified range are displayed in red.

	Element	Cross sections	Strength/stability work ratio	Deflection work ratio	Suggested solutions	Strength/stability work ratio	Deflection work ratio	Accepted solutions
	4	PE200	79 %	33 %	IPE200	79 %	33 %	
	5	HEA200	52 %	53 %	HEA200	52 %	53 %	
	9	HEA200	103 %	73 %	HEA220	77 %	50 %	HEA220
	10	PE200	159 %	34 %	IPE220	98.%	24 %	IPE220
	14	HEA200	103 %	73 %	HEA220	77 %	50 %	HEA220
	15	PE200	159 %	34 %	PE220	98 %	24 %	IPE220
tion method	19	HEA200	52 %	53 %	HEA200	52 %	53 %	
and i meet now	20	IPE200	79 %	33 %	IPE200	79 %	33 %	
emerx	21	HEA200	56 %	21%	HEA200	56 %	21 %	
ection	22	HEA200	59 %	11 %	HEA200	59 %	11 %	
esign template	23	IPE200	45 %	47 %	IPE200	46 %	47 %	
me	24	IPE200	56 %	55 %	IPE200	56 %	55 %	
	25	IPE200	59 %	59 %	IPE200	59 %	59 %	
yaxem	26	IPE200	59 %	59 %	IPE200	59 %	59 %	
and all	27	IPE200	48 %	47 %	IPE200	46 %	47 %	
Accept at	28	IPE200	56 %	55 %	IPE200	56.%	55 %	
Reject all								

Optimization of steel shapes

- 2. Click **Accept all** to accept the suggested shapes.
- 3. Click **<OK>** to close and apply.
- 4. Rerun the finite element calculation and the steel calculation.

After the calculation, open the 'Suggested shapes' dialog box. If there are other suggested shapes, repeat the above steps until you obtain the work ratio comprised in the specified range for all steel shapes.

Element	Cross sections	Strength/stability work ratio	Deflection work ratio	Suggested solutions	Strength/stability work ratio	Deflection work ratio
4	IPE200	77 %	32 %	IPE200	77 %	32 %
5	HEA200	51 %	52 %	HEA200	51 %	52 %
9	HEA220	90 %	62 %	HEA220	90 %	62 %
10	IPE270	91 %	23 %	IPE270	91 %	23 %
14	HEA220	90 %	62 %	HEA220	90 %	62 %
15	IPE270	91 %	23 %	IPE270	91 %	23 %
19	HEA200	51 %	52 %	HEA200	51 %	52 %
20	IPE200	77 %	32 %	IPE200	77 %	32 %
21	HEA200	53 %	20 %	HEA200	53 %	20 %
22	HEA200	58 %	11 %	HEA200	58 %	11 %
23	IPE200	44 %	44 %	IPE200	44 %	44 %
24	IPE200	50 %	49 %	IPE200	50 %	49 %
25	IPE200	52 %	51 %	IPE200	52 %	51 %
26	IPE200	52 %	51 %	IPE200	52 %	51 %
27	IPE200	44 %	44 %	IPE200	44 %	44 %
28	IPE200	50 %	49 %	IPE200	50 %	49 %

> 8.2.4 Saved post-processing views

A post-processing view saves the entire post-processing scenario (i.e., result type, result component, selected analyses and elements, results in visualization settings) together with the display settings of the model (i.e., viewpoint, rendering, etc.). For each post-processing view, a corresponding image file is saved to disk. The saved images are in the **Document** mode of the project browser.

Post-processing views automatically replay the saved post-processing, without having to manually recreate the post-processing scenario. Moreover, if the structure settings were changed and the results were modified, the updated post-processing views display the new results.

Example: Creating a post-processing view

1. Define a (1, -1, 1) view of the workplane: press <Alt + 5>.

- 2. Set a ghost display of the descriptive model: on the **Home** tab, **Render Modes** panel, click 💷 .
- 3. Access the 'Results' dialog box by pressing **<Alt + Z>**. On the **FE** tab, select:
 - Forces result type.
 - My result on linear elements.
 - Mxx result on planar elements.
 - 101 loads combination.
- 4. Click **<OK>**.
- 5. On the **Results** tab, click **Save View** to save the post-processing.



Creating a post-processing view-My efforts

To access a saved post-processing view: double-click it in the Project Browser.

> 8.2.5 Reports

Advance Design provides an advanced and powerful report generator tool, with which the desired reports are easily defined. The available report templates can be used, or new templates can be defined. The report generator filters its content according to the current settings and available results. At the same time, the report content takes the selection of elements (if any) into account.

Example: Generating a report

1. From the menu, select **Documents > Standard Report**; the report generator automatically loads the standard report template.

Report Generator	? ×
Image: Standard Report Image: Standard Report Standard Report Image: Standard Report Standard Report Image: Standard Report Standard Rep	Document Table Post-processing Curve Views Other A Cover Sheet Table of contents Ch Chapter Text Text Mage Section break Page break
Load Save Table properties Case/Combinations	X Create tables
	Display details Generate Cancel Close

Generating a report

2. On the **Post-processing** tab of the report generator, select the post-processing view.

3. In the report's content section, select the table with forces on planar elements and click the arrows to insert the post-processing right below.

Report Generator	? ×
Report Generator	? × Document Table Post-processing Curve Views Other
Load Save Table properties Case/Combinations	Create tables
	Display details Generate Cancel Close

4. Click **Case / Combinations** and select only combination **101** in the **Displacements**, **Forces**, and **Stresses** tabs.

5. Click **Generate** to start the report creation. When finished, the report is automatically displayed with the document viewer application.



9. Design Modules

The Design mode is used for designing reinforced concrete elements and steel connections using the Advance Design Reinforced Concrete and Steel Connection modules. The Design mode is only available after the FEM analysis is completed.

The Design Modules ribbon only becomes active after analysis is performed, and an element is selected in the Design tab of the Project Browser by double-clicking on it.

The Design Modules tab of the Advance Design Ribbon provides you with access to commands from the Reinforced Concrete module and Steel Connection module. This way, you can achieve the entire design process within Advance Design.



The Design Modules tab of the Advance Design Ribbon

9.1 Concrete Elements

Once the FEM analysis is complete, the user can open any element from the model by:

1. Applying the Open with Design Modules command from the contextual menu



2. By double-clicking on an element in the **Design** tab of the Project Browser.



Five types of concrete elements can be calculated using the Advance Design modules: footings (continuous and isolated), beams, columns, slabs, and walls.





Advance Design modules: isolated footing



Advance Design modules: beam



Advance Design modules: column



Advance Design modules: slab





C

The unavoidable step before running a structural design session is to define the structural design assumptions.

Structural design assumptions can be split into two different families:

• Design assumptions (Design Settings)

C Design Assumptions		×
	Footing Reinforcement	
General Design	Design method	Strut-and-tie ~
Footing Design	Consider earth loads and overloads	
Bearing Verification Geotechnical Verifications	Consider footing self-weight	\checkmark
Multilayered Soil Verification	Provide minimum reinforcement area	\checkmark
Seismic Design Reinforced Concrete	Provide constructive reinforcement area	Птор
Cracking		Bottom
Concrete Covers	Constructive reinforcement area	160.00 mm²
	Pedestal Reinforcement	
	Consider pedestal self-weight	
	Reinforcement increment step	10.00 mm ²
	Preliminary Sizing	
	dA	0.05 m
	dB	0.05 m
	dH	0.05 m
	Number of iterations	10
ADVANCE DESIGN	1	Ok Apply Close 💕

Design Assumptions

• Constructive dispositions for the automatic placement of bars inside concrete members (Reinforcement Settings)

C Reinforcement Assumptions			×
	General Settings		
·• •	Reinforcement along X	Bars with hooks	
Pad		Closed links	
Supported Element	Reinforcement along Y	Bars with hooks	F
 Longitudinal 		Closed links	
Transversal	Distribution type	Standard v	
	Minimum bars spacing	0.04 m	
	Maximum bars spacing	0.30 m	
	Uniform spacing	0.00 m	
	Maximum bars diameter	ø10 v	
	Provide welds for transversal bars		
	Hooks		
	Angle	Auto v	
	Bottom bars along X / Y	✓ / ✓	
	Top bars along X / Y	✓ / ✓	
	Links hooks angle	135° 📉 🗸	
	Mandrel Diameters for Bent Bars		
	Method	From library v	
	List of available diameters	30 40 60 80 100 125 160 20	
ADVANCE DESIGN			Ok Apply Close 🕼

Reinforcement Assumptions

Both assumption families (Design and Reinforcement) are country-code dependent (Eurocodes with different national appendixes, US codes, Canadian codes, NTC codes).

When all assumptions are set, you can calculate the model. The modules will automatically generate the reinforcement needed for the efforts transferred from Advance Design.



Presentation of Main and Secondary bars

After the reinforcement is generated, you still have the option to adjust it. The reinforcement can easily be modified, or new bars can be added, with the possibility to see if the area of modified bars exceeds the theoretical area calculated by the Advance Design module.

The calculated reinforcement can be restored at any point by using the 'Restore from calculation' option.

C Reinforcement				X
	Reinforcement Sections			Q
	Theoretical reinforcement		300.10 mm ²	→━╋━┥━╋━╴┲
A Pad	Real reinforcement		314.00 mm ²	S
Bottom Bars Along Y	General Settings			s s
Top Bars Along X	Bar mark		1	→
Top Bars Along Y Supported Element Longitudinal 	Bar color		<u> </u>	→ → S
Supported Element Transversal	Quantity	(Q):	4	s s
	Diameter		ø10 ~	, [™]
	Spacing	(S):	28.67 cm	L,×
	Offset	(O):	7.00 cm	
	Covers			
	Тор	(T):	0.00 cm	
	Bottom	(B):	7.50 cm	14 12
	Lateral left	(L1):	4.00 cm	
	Lateral right	(L2)	4.00 cm	
	Hooks			
	Angle left		90.00 * ~	1 <u>+</u> B 2
	Angle right		90.00 * ~	
	Length left		0.16 m	
	Length right		0.16 m	
	Restore from calculation		Restore	
ADVANCE DESIGN				Ok Apply Close 🕼

Reinforcement dialog - 'Restore from calculation' option

The status of calculation results is displayed in the Design mode for each separate element after performing the calculation. A green check sign will appear for the calculated element that has no error message or failed verifications, which means that the obtained maximum work ratio for that element is smaller than 100%. Otherwise, when at least one verification provides a work ratio greater than 100% or at least one error message is displayed, an X red sign will appear.



9.2 Steel Connections

The steel connections available in Advance Design make it possible to quickly design a variety of connections in Advance Design projects. The Steel Connection module will help you work more productively when modeling, covering all the steps needed to define a steel joint and customize its properties including the main members of the connection, additional joint elements, connectors, and materials.

Once the element is opened in the module (in the same way as for concrete elements), the joint elements and calculation assumptions can be modified to satisfy your needs.



Steel Connection - Geometry dialog

The efforts from Advance Design are transferred as the envelope of efforts for a simple reason: for steel structure, there are too many load cases\combinations and the calculations take too long.

Using the envelope, the calculation is done in just a few seconds.

C Steel Connection - Loads and combinations X									
Load Cases Combi	nations	; Lo	oads d	lefinition					
Column	+	19			~ →				
	Add	Del	ID	Code	Load case		V	М	N
	+	×	1	ECG	1 - D	~	8.85 kN	0.00 kN⋅m	-13.95 kN
	+	×	2	ECQ	2 - L	Š	0.00 kN	0.00 kN⋅m	0.00 kN
	+	×	3	ECN	3 - Snw	Ś	7.09 kN	0.00 kN⋅m	-7.17 kN
	+	×	4	ECV	4 - WX+S	~	-32.18 kN	0.00 kN⋅m	33.86 kN
	+	×	5	ECN	5 - SndrftX+	ç	4.84 kN	0.00 kN⋅m	-2.90 kN
	+	×	6	ECN	6 - SndrftX-	~	7.08 kN	0.00 kN⋅m	-12.65 kN
	+	×	7	ECN	7 - SndrftY+	~	8.43 kN	0.00 kN⋅m	-8.78 kN
	+	×	8	ECN	8 - SndrftY-	~	7.18 kN	0.00 kN⋅m	-7.51 kN
	+	×	9	ECV	9 - WX+S2	~	-18.35 kN	0.00 kN⋅m	11.17 kN
	+	×	10	ECV	10 - WX-S	~	0.54 kN	0.00 kN⋅m	9.04 kN
	+	×	11	ECV	11 - WX-S2	~	14.55 kN	0.00 kN⋅m	-6.48 kN
	+	×	12	ECV	12 - WY+S	~	-9.79 kN	0.00 kN⋅m	70.93 kN
	+	×	13	ECV	13 - WY+S2	~	-9.72 kN	0.00 kN⋅m	71.02 kN
	+	×	14	ECV	14 - WY-S	~	-3.19 kN	0.00 kN⋅m	-29.56 kN
	+	×	15	ECV	15 - WY-S2	~	-3.04 kN	0.00 kN⋅m	-31.04 kN
+ x 15 ECV 15 - WY-S23.04 kN 0.00 kN·m31.04 kN EEE ECV 15 - WY-S23.04 kN 0.00 kN·m31.04 kN EEE ECV ERSET all loads values									
ADVANCE DESIG	N							Ok Ap	ply Cancel

Steel Connection – Loads and combinations

You can know exactly from which combinations from Advance Design the envelopes are created because in front of each envelope the combination is displayed. At any time, supplementary efforts can be added.

The status of calculation results is displayed in the Design mode for each separate connection after performing the calculation. A green check sign will appear for the calculated connection that has no

error message or failed verifications, which means that the obtained maximum work ratio for that element is smaller than 100%. Otherwise, when at least one verification provides a work ratio greater than 100% or at least one error message is displayed, an X red sign will appear.



9.3 Output: Calculation Results Tab

For each performed verification, the 'Calculation results' tab displays the verification type, combination, value, limit, and work ratio.

Verification type	Combination	Value Limit		Work Ratio
Bearing resistance	aring resistance 116: 1.35x[1 G]+1.5x[2 Q]+0.75x [3 N]		363.08 kN	108.47%
Sliding	113: 1x[1 G]+1.5x[2 Q]	38.32 kN	130.02 kN	29.47%
Overturning	Overturning 101: 0.9x[1 G]		1.50	48. 73%
Settlement	123: 1x[1 G]+1x[2 Q]+0.5x[3 N]	11.09 cm 5.00 cm		221.76%
Punching	116: 1.35x[1 G]+1.5x[2 Q]+0.75x [3 N]	0.37 MPa	0.78 MPa	47. 28%
Reinforcement	Real	Theoretical		Ratio
Bottom Along X	1020.50 mm²	945.70 mm²		92.67%
Bottom Along Y	628.00 mm ²	550.7	1 mm²	87.69%
Top Along X	352.10 mm²	312.1	0 mm²	88.64%
Top Along Y	301.80 mm²	78.03	mm²	25.85%

Calculation results

Once the element is calculated, you can display the drawing on the screen, print it, or save the drawing on the computer.

On the drawing, the program automatically creates a bar schedule and a title block with all the information about the project and the element (this information will be imposed in 'Drawing Settings' from Advance Design modules).



Interactive Drawings - Rigid Point Support

You can manage all drawing components in the same window. For example, you can quickly add any number of new views to a given drawing.

9.4 Advance Design Modules Reports

One of the greatest features of Advance Design modules is producing a set of available reports. You can preview reports using the Report Designer tool or export them in DOC or PDF format.



Report Designer tool

You can define general options regarding calculation reports such as report type, chapter customization, or saving options (format, location). These settings will be further considered when launching the reports generation or preview commands.



Report presentation

9.5 Masonry Wall Module

Using the Advance Design Masonry Wall module, it is possible to design masonry walls, modeled as planar elements (shell elements).

> 9.5.1 Workflow

A masonry wall modeled in Advance Design must meet several conditions to be exported/opened in the Masonry Wall module:

- Vertical planar element,
- Material: masonry,
- Shape: rectangular,
- With/without openings.

Please note that the masonry material contains information about the masonry construction (including the number and type of layers, their thickness, type of masonry units, and mortar) and can be created directly or imported from the **Libraries.** Depending on the norm selected in the **Localization** settings, different types of materials can be selected. Six types of walls (by the number and type of layers) are available and can be selected depending on your needs.

GRAITEC

			Materia	als			7
	Designation	Family	Standard	Туре	Color		
	S235	STEEL	EN 10025-2	S235	8c		
	S275	STEEL	EN 10025-2	S275	b0		
	S355	STEEL	EN 10025-2	S355	e6		
	C25/30	CONCR	EN206	C25/30	6c		
	C20/25	CONCR	EN206	C20/25	58		
	BOISC	TIMBER	Timber User	User	00		
₽	Single-leaf Wall	MASON	EN 1996/RO	Single-leaf	00		
ibr	ary C:\ProgramData\G	raitec\Advan	ce Design\2023\R	lesources\Catal	ogs\Mater	ial.ac 💌	6 6 0
ibr Fa	rary C:\ProgramData\G mily MASONRY	raitec\Advan	ce Design\2023\R 🔹 Standard	esources\Catal	ogs\Mater ieral	ial.ac 💌	
ibr Fai Do Fai Stif	ary C:\ProgramData\G mily MASONRY vity wall uble-leaf wall ced Wall ounted cavity wall igle-leaf Wall fened single-leaf wall	raitec\Advan , all	ce Design\2023\F	tesources\Catal	ogs\Mater	ial.ac 🔪	 Import Export Delete

Material libraries

Please note that the material properties (masonry unit, mortar, thickness, etc.) will be defined in the same window, depending on the selected standard:

Designation Family Standard Type Color S235 STEEL EN 10025-2 \$235 8c S275 STEEL EN 10025-2 \$235 6c S355 STEEL EN 10025-2 \$355 6c C25/30 CONCR EN206 C25/30 6c C20/25 CONCR EN206 C20/25 58 BOISC TIMBER Timber User User 00 Single-leaf Wall MASON. EN 1996/RO Single-leaf 00 fechanical Properties >> Properties EN 1996/RO Libraries >> Libraries >> section name Single-leaf Wall * * type Single-leaf Wall * * Masony unit Solid Clay Dirck, class 10 Leaf 2 (inner) Motar General purpose - M1 Thickness 20.00 cm Fully filed vertical joints Yes East 10 East 10 Motar General purpose - M1 Thickness 20.00 cm Fully filed vertical joints Yes East 10 Thickness<						
S235 STEEL EN 10025-2 S235 8c S275 STEEL EN 10025-2 S275 b0 S355 STEEL EN 10025-2 S275 b0 C25/30 CONCR EN206 C25/30 6c C20/25 CONCR EN206 C20/25 58 BOISC TIMBER Timber User User 00 Single-leafWall MASON EN 1996/RO single-leaf 00 Add Delete Pure Single-leaf Wall * Libraries >> Single-leaf Wall * Masonry unit Solid Clay Drick, class 10 Mortar General purpose - M1 Thickness 20.00 cm </th <th>Designation</th> <th>Family</th> <th>Standard</th> <th>Туре</th> <th>Color</th> <th></th>	Designation	Family	Standard	Туре	Color	
S275 STEEL EN 10025-2 S275 b0 S355 STEEL EN 10025-2 S355 6 C25/30 CONCR EN206 C25/30 6c C20/25 CONCR. EN206 C20/25 58 BOISC TIMBER Timber User User 00 Single-leaf Wall MASON. EN 1996/RO Single-leaf 00 Single-leaf Wall Delete Pur Add Delete Pur Add Delete Pur Add Delete Pur Add Delete Pur Single-leaf Wall * type Single-leaf wall * Single-leaf Wall * Masony unt Solid Clay brick, class 10 Motar General purpose - M1 Thickness 20:00 cm Fully filled vertical joints Yes Bearing Yes Leaf Delete Pur Single-leaf Vall * Cart thickness 20:00 cm Fully filled vertical joints Yes Bearing Yes Leaf Delete Pur Could Clay Drick, class 10 Motar General purpose - M1 Thickness 20:00 cm Fully filled vertical joints Yes Bearing Yes Leaf Delete Pur Could Clay Drick, class 10 Motar General purpose - M1 Thickness 20:00 cm Fully filled vertical joints Yes Bearing Yes Leaf Delete Pur Could Clay Drick, class 10 Thickness 20:00 cm Thickness 20:00 cm Thickness 20:00 cm Cavity thickness 20:00 cm Thickness 20:00 cm	S235	STEEL	EN 10025-2	S235	8c	
S355 STEEL EN 10025-2 S355 6. C25/30 CONCR. EN206 C25/30 6c. C20/25 CONCR. EN206 C20/25 58. BOISC TIMBER TimberUser User 00. Single-leafWall MASON. EN 1996/RO Single-leaf 00. Single-leafWall Delete Pur Add Delete Pur Add Delete Pur Add Delete Pur Single-leafWall * Elibraries >> Section name Single-leaf Wall * type Single-leaf Wall * Solid Clay brick, class 10 Motar General purpose - M1 Thickness 20.00 cm Fully filled vertical joints Yes Bearing Yes Leaf 2 (inner) Solid Clay brick, class 10 Motar General purpose - M1 Thickness 20.00 cm Fully filled vertical joints Yes Bearing Yes Leaf 2 (inner) Leaf 2 (inner) Leaf 2 (inner) Leaf 2 (inner) Control Clay Dick Class 10 Motar General purpose - M1 Thickness 20.00 cm Fully filled vertical joints Yes Bearing Yes Leaf 2 (inner) Control Clay Dick Class 20.00 cm Thickness 20.00 Cm Solid Clay Dick Class 10 Total thickness 20.00 cm Thickness 20.00 cm Thickness 20.00 cm	S275	STEEL	EN 10025-2	S275	ь0	
C25/30 CONCR. EN206 C25/30 6c C20/25 CONCR. EN206 C20/25 58 BOISC TIMBER Timber User User 00 Single-leafWall MASON. EN 1996/RO Single-leaf 00 echanical Properties >> Properties EN 1996/RO Libraries >> Libraries >> section name Single-leaf Wall type Single-leaf Wall Masonry unit Solid Clay brick, class 10 Leaf 2 (inner) Motar General purpose - M1 Thickness 20.00 cm Fully filled vertical joints Yes Bearing Yes Leaf parameters Main 0.00 MPa fm 0.00 MPa Total thickness 20.00 cm Leaf parameters 20.00 cm Thickness connected by ties 20.00 cm <	S355	STEEL	EN 10025-2	S355	e6	
C20/25 CONCR. EN206 C20/25 58 BOISC TIMBER Timber User User 00 Single-leafWall MASON EN 1996/RO Single-leaf 00 Add Delete Pur echanical Properties >> Properties EN 1996/RO Libraries >> section name Single-leaf Wall • type Single-leaf Wall • Masonry unit Solid Clay brick, class 10 • Mortar General purpose - M1 • Fully filled vertical joints Yes • Bearing Yes • Leaf parameters ✓ Automatically calculated • fm 0.00 MPa • k 2.75 MPa • • Longitudinal joint No • • • Layers connected by ties 20.00 cm • Thickness for bearing 20.00 cm	C25/30	CONCR	EN206	C25/30	6c	
BOISC TIMBER Timber User User 00 Single-leaf Wall MASON EN 1996/RO Single-leaf 00 Add Delete Pur echanical Properties >> Properties EN 1996/RO<	C20/25	CONCR	EN206	C20/25	58	
Single-leaf Wall MASON EN 1996/RO Single-leaf 00 Add Delete Pur Add Delete Pur echanical Properties >> Properties EN 1996/RO Libraries >> section name Single-leaf Wall type Single-leaf Wall Wasonry unit Solid city brick, class 10 Leaf 2 (inner) Masonry unit Solid city brick, class 10 Leaf 2 (inner) Motar General purpose - M1 Thickness Thickness 20.00 cm Eaf parameters Matomatically calculated Delete Pur b 10.00 MPa Eaf parameters Longitudinal joint No Total thickness 20.00 cm Thickness connected by ties Total thickness 20.00 cm	BOISC	TIMBER	Timber User	User	00	
Add Delete Pur echanical Properties >> Properties EN 1996/RO<	Single-leaf Wall	MASON	EN 1996/RO	Single-leaf	00	
Add Delete Pur echanical Properties >> Properties EN 1996/RO Libraries >> section name Single-leaf Wall • type Single-leaf wall • Masonry unit Solid clay brick, class 10 Leaf2 (nner) Masonry unit Solid clay brick, class 10 Leaf2 (nner) Mortar General purpose - M1 Thickness Fully filled vertical joints Yes Searing yes						
achanical Properties >> Properties EN 1996/R0 Libraries >> section name Single-leaf Wall ype Single-leaf wall value Leaf1 (outer) Leaf2 (inner) Wasonry unit Solid clay brick, class 10 Leaf2 (inner) Wasonry unit Solid clay brick, class 10 Leaf2 (inner) Wasonry unit Solid clay brick, class 10 Leaf2 (inner) Wasonry unit Solid clay brick, class 10 Leaf2 (inner) Wasonry unit Solid clay brick, class 10 Leaf2 (inner) works 20.00 cm Early properties uitly filled vertical joints Yes Early calculated b 10.00 MPa Early calculated b 10.00 MPa Early calculated wayers connected by ties Total thickness 20.00 cm Cavity thickness 20.00 cm Thickness for bearing				Ad	d De	lete Pu
echanical Properties >> Properties EN 1996/R0< Libraries >> Libraries => Librar						
Section name Single-leaf Wall type Single-leaf Wall Wasonry unit Solid Clay brick, class 1U Wasonry unit Solid Clay brick, class 1U Wortar General purpose - M1 Thickness 20.00 cm Fully filled vertical joints Yes Bearing Yes Leaf parameters ✓ Automatically calculated b 10.00 MPa k 2.76 MPa	echanical Prope	ties>>	Properties	N 1996/RO <<		Libraries >>
section name Single-leaf Wall Single-leaf Wall Single-leaf Wall U Single-leaf Wall U Solid Clay brick, class 10 Mortar General purpose - M1 Thickness 20.00 cm Uly filled vertical joints Yes Searing Yes seaf parameters Automatically calculated b 10.00 MPa k 2.76 MPa Congitudinal joint No Total thickness 20.00 cm Thickness for bending 20.00 cm Thickness for bending 20.00 cm Thickness for bending 20.00 cm	of an and a second second	100	r topenico i			cibrandorr
Masonry unit Solid clay brick, class 10 Mortar General purpose - M1 Thickness 20.00 cm Fully filled vertical joints Yes Bearing Yes Leaf parameters Automatically calculated th 10.00 MPa fm 0.00 MPa fk 276 MPa Longitudinal joint Layers connected by ties Cavity thickness	section name type		Single-leaf Wal		•	
Montar General purpose - M1 Thickness 20.00 cm Thickness 20.00 cm Huly filled vertical joints Yes Leaf parameters Automatically calculated th 10.00 MPa fm 0.00 MPa fk 2.76 MPa Longitudinal joint Layers connected by ties Cavity thickness 0.00 cm Thickness for bearing 20.00 cm	section name type		Single-leaf Wal Single-leaf wall Leaf 1 (oute	i)	▼ Leaf2(in	iner)
Inickness 20.00 cm Fully filled vertical joints Yes Bearing Yes Leaf parameters Automatically calculated tb 10.00 MPa fm 0.00 MPa fk 2.76 MPa Longitudinal joint Layers connected by ties Cavity thickness Cavity thicknes Cav	section name type Masonry unit		Single-leaf Wal	í) ICK, class IU	Leaf2 (in	iner)
Fully filed vertical joints Yes Bearing Yes Leaf parameters Automatically calculated tb 10.00 MPa fm 0.00 MPa fk 2.76 MPa Longitudinal joint Layers connected by ties Cavity thickness Cavity thicknes Cavity	section name type Masonry unit Mortar		Single-leaf Wall Single-leaf wall Leaf 1 (oute Solid clay bi General pur	r) ick, class 10 bose - M1	Leaf2 (in	iner)
Leaf parameters balanger ves Leaf parameters b 10.00 MPa fm 0.00 MPa fk 2.76 MPa Longitudinal joint Layers connected by ties Cavity thickness	section name type Masonry unit Mortar Thickness		Single-leaf Wall Single-leaf wall Leaf 1 (oute Solid clay bi General pur 20.00 cm	i) ICK, class IU Doose - M1	Leaf2 (in	iner)
Longitudinal joint Layers connected by ties Cavity thickness	section name type Masonry unit Mortar Thickness Fully filled vertica	ljoints	Single-leaf Wall Single-leaf wall Leaf 1 (oute Solid clay bu General pun 20.00 cm Yes	i) ick, class IU pose - M1	Leaf2(in	iner)
In the second se	section name type Masonry unit Mortar Thickness Fully filled vertica Bearing	ljoints	Single-leaf Wal Single-leaf wall Leaf 1 (oute Solid Clay bi General pun 20.00 cm Yes Yes) ick, class 10 pose - M1	Leaf2(in	iner)
In the second se	section name type Masonry unit Mortar Thickness Fully filled vertica Bearing Leaf parameters &	l joints	Single-leaf Wal Single-leaf wall Solid clay bi General pun 20.00 cm Yes Yes Automat) ick, class 10 joose - M1 cally calculated	Leaf2 (in	iner)
Longitudinal joint Layers connected by ties Cavity thickness	section name type Masonry unit Mortar Thickness Fully filled vertica Bearing Leaf parameters th	ljoints	Single-leaf Wal Single-leaf Wal Solid Clay bi General pun 20.00 cm Yes Yes Yes Automat 10.00 MPa) ick, class 10 ics - M1 cally calculated	Leaf2 (in	iner)
Longitudinal joint Layers connected by ties Cavity thickness	section name type Masonry unit Mortar Thickness Fully filled vertica Bearing Leaf parameters fb fm m	ljoints	Single-leaf Wal Single-leaf wall Leaf 1 (oute Solid clay bi General pun 20.00 cm Yes Yes Automat 10.00 MPa 0.00 MPa	i) ick, class 10 pose - M1 cally calculated	• Leaf2 (in	iner)
Layers connected by ties Cavity thickness	section name type Masonry unit Mortar Thickness Fully filled vertica Bearing Leaf parameters fb fm fk	ljoints	Single-leaf Wall Single-leaf wall Leaf 1 (oute Solid Clay bi General pun 20.00 cm Yes Yes Xes Automat 10.00 MPa 2.76 MPa	r) rck, class 10 pose - M1 ically calculated	v Leaf2 (in	iner)
Cavity thickness 20.00 cm	section name type Masonry unit Mortar Thickness Fully filled vertica Bearing Leaf parameters fb fm fk	l joints	Single-leaf Wall Single-leaf wall Leaf 1 (oute Solid Clay bi General pun 20.00 cm Yes Yes Automat 10.00 MPa 2.76 MPa) ick, class 10 oose - M1 cally calculated	Leaf2 (in	iner)
Thickness of bearing 20.00 cm	section name type Masonry unit Mortar Thickness Fully filled vertica Bearing Leaf parameters fb fm fk Longitudinal joint	l joints No	Single-leaf Wall Single-leaf wall Leaf 1 (oute Solid Clay bi General pun 20.00 cm Yes Yes Yes Automat 10.00 MPa 2.76 MPa) ick, class 10 oose - M1 cally calculated	Leaf2 (in	20.00 cm
Covity infil	section name type Masonry unit Mortar Thickness Fully filled vertica Bearing Leaf parameters fb fm fk Longitudinal joint Layers connecter	l joints	Single-leaf Wall Single-leaf wall Leaf 1 (oute Solid clay bi General pun 20.00 cm Yes Yes Yes Automat 10.00 MPa 2.76 MPa) ick, class 10 ics - M1 ically calculated	Leaf2 (in Leaf2 (in ckness ss for bending us for bending	20.00 cm 20.00 cm 20.00 cm
	section name type Masonry unit Mortar Thickness Fully filled vertica Bearing Leaf parameters fb fm fk Longitudinal joint Layers connecte Cavity thickness Cavity thickness	I joints	Single-leaf Wall Single-leaf wall Leaf 1 (oute Solid Clay bi General pun 20.00 cm Yes Yes Yes Automat 10.00 MPa 2.76 MPa) ick, class 10 cose - M1 cally calculated Total thic Thickney	Leaf2 (in Leaf2 (in ckness ss for bending ss for bearing	20.00 cm 20.00 cm

Properties



The workflow is the same as for reinforced Concrete walls: after running the FEM analysis, masonry walls can either be opened directly, using the Design tab or can be exported to the standalone module, using a command from the right-click menu.



Opening/exporting a masonry wall in the Design Module

The following data will be exported to the Masonry Wall module:

- Wall Geometry
- Openings
- Loads:
 - Load cases and combinations
 - FEM results (efforts)

In the Masonry module, you can modify the wall geometry, wall section, and design assumptions before running the calculation.



Masonry wall module

> 9.5.2 Calculations and results

The Advance Design Masonry Wall module supports calculations according to Eurocode 6 - with national appendixes, the Italian norm NTC 2018, and the Romanian norm CR6-2013.

Note: The National appendix for Germany is not available in version 2023.

Depending on the selected norm, the Masonry Module performs a Detailed or Simplified Calculation, and a Calculation report will be available.

Geometry description 1





3	Global assumptions				
Mase	onry design calculation:	EN1996-1-1/EN 1996-3.			
Load	lings and combinations:	IBC 2012.			
Cale	ulation method:	Detailed			
Fire	data				
Fire	resistance:	No fire			
Crite	ria:	R			
Expo	sed sides:	1			
3.1	Localization				
Loca	lization	Europe			
Elen	ent type	Masonry Wall			
Elem	ent ID	1			
Posit	ion	MasonryWall 1, Level 1			
Drav	ving	M			
Leve	1				
3.2	Units				
Leng	ths	mm			
Forc	es	kN			
Moments		kN · m			
Stresses		MPa (N/mm ²)			
Angles		0			
All ti	he lengths are linked to the	"Small lengths" unit from the GUI			
37.37					

3.3 Material

3

Partial Factors 7 _M						
	ULS		SLS	AULS	SULS	
Compression	Shear	Flexural Tension				
2	2	2	1	2	1.5	

Verification of wall loaded mainly vertically (EN 1996-1-1 - Detailed method) 4

General calculation data 4.1

Clear height of wall Clear height of wall

Thickness of the upper slab / beam

Height of the level

$$\begin{split} h &= h_L - D_t \\ h &= 3000.0 \text{ mm} - 200.0 \text{ mm} = 2800.0 \text{ mm} \\ h_L &= 3000.0 \text{ mm} \end{split}$$
 $D_1 = 200.0 \text{ mm}$

Calculation report

Please click on the links below for detailed information about the Advance Design Modules' capabilities and features:

- Advance Design Steel Connection module •
- Advance Design RC Footing module •
- Advance Design RC Beam module •
- Advance Design RC Wall module ٠
- Advance Design Column module •
- Advance Design Masonry Wall •