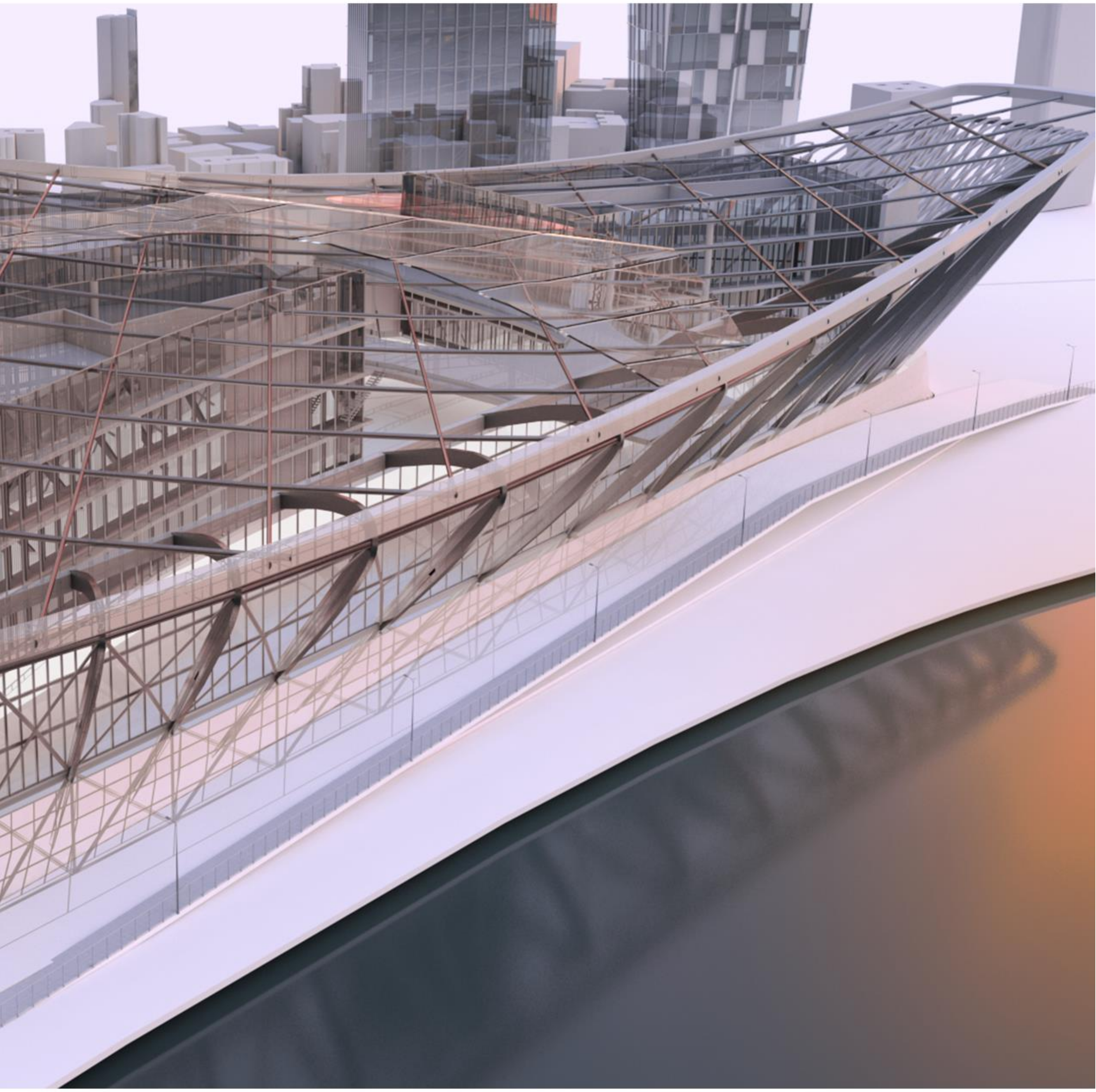
ADVANCE DESIGN 2022

Starting Guide



GRAITEC

Table of contents

Introduction

Welcome to Advance Design

From modeling to the structure calculation, result post-processing and structure optimization, Advance Design offers a complete environment for the static and dynamic analysis of 2D and 3D structures using the finite elements 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 analysis software, improved with powerful and innovative features:

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

About this Guide

This guide provides a description of 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 of Advance Design and not all its features are described. For detailed information regarding the program functions, refer to the Advance Design Help.

Technical Support

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

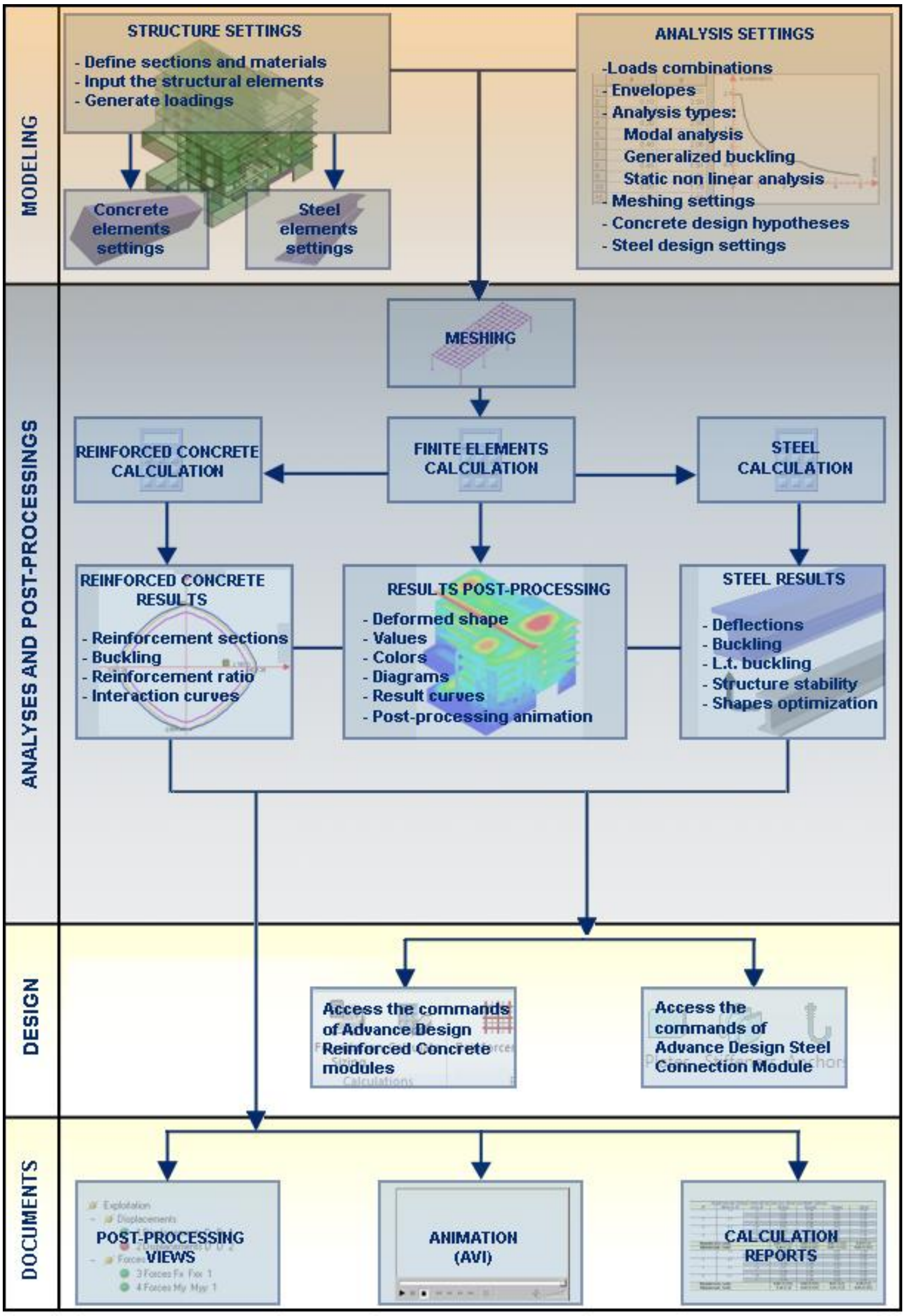
- **Ribbon**: **Manage** tab: click **Technical support** to send an email to GRAITEC.
- **Menu**: select **Help** > **Technical support**. A template email is sent to the technical support team who will quickly solve the problem and provide a precise answer. The model of the current project is automatically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

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**.



Installing Advance Design

System Requirements

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

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 Graitec Advantage 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 on launching Advance Design. Follow the procedure described in the Installation guide.

Starting Advance Design

Advance Design can be launched using various methods:

- From the Windows Start menu, select Programs > Graitec > Advance Design.
- Double click the Advance Design icon on the desktop.
- 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.

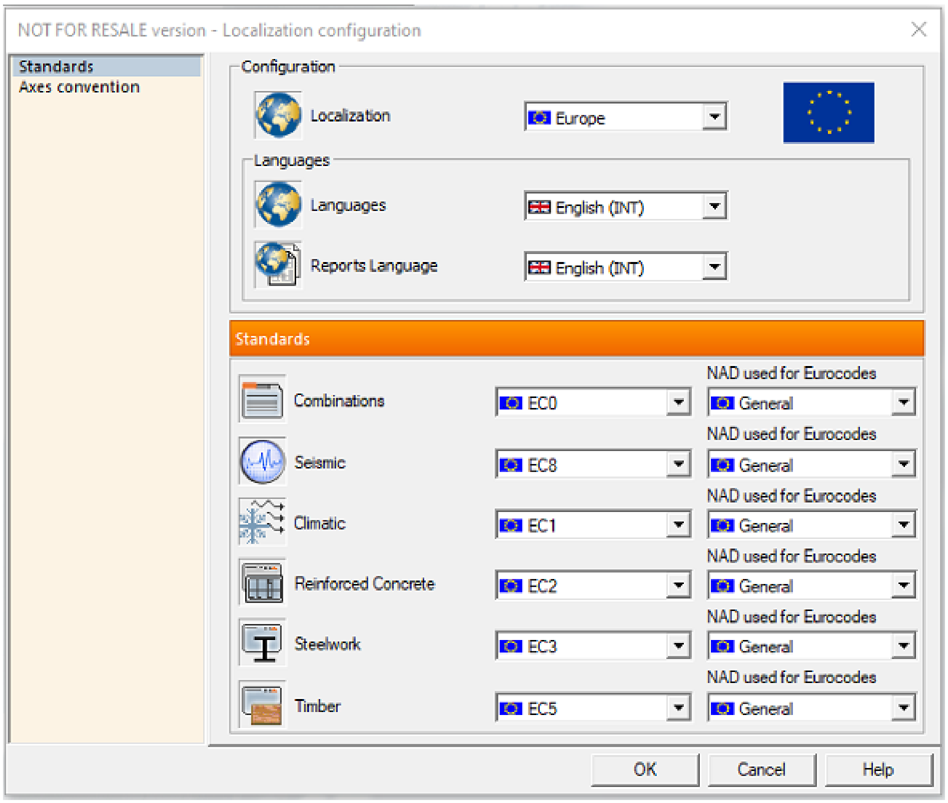
Project Management

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.

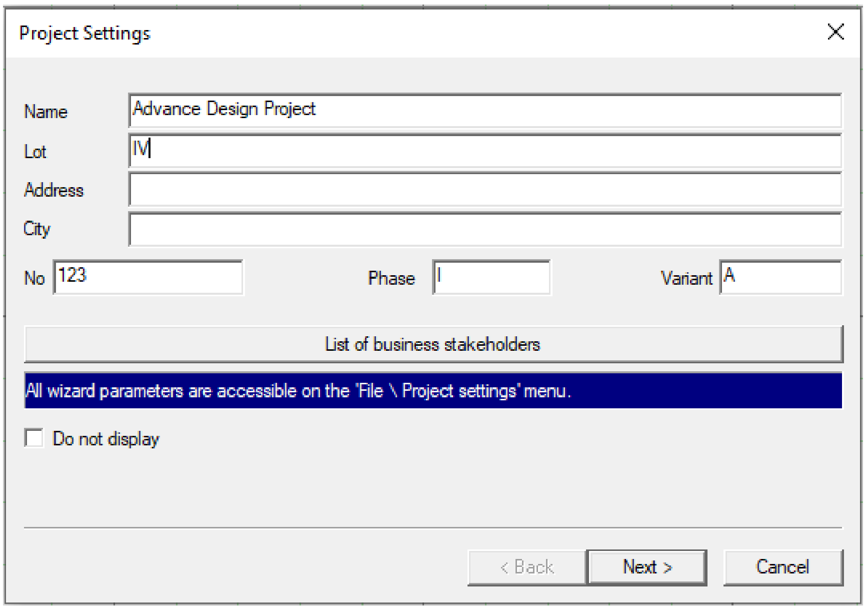
The Advance Design start page has a left side panel and two slides: CREATE and LEARN.

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.

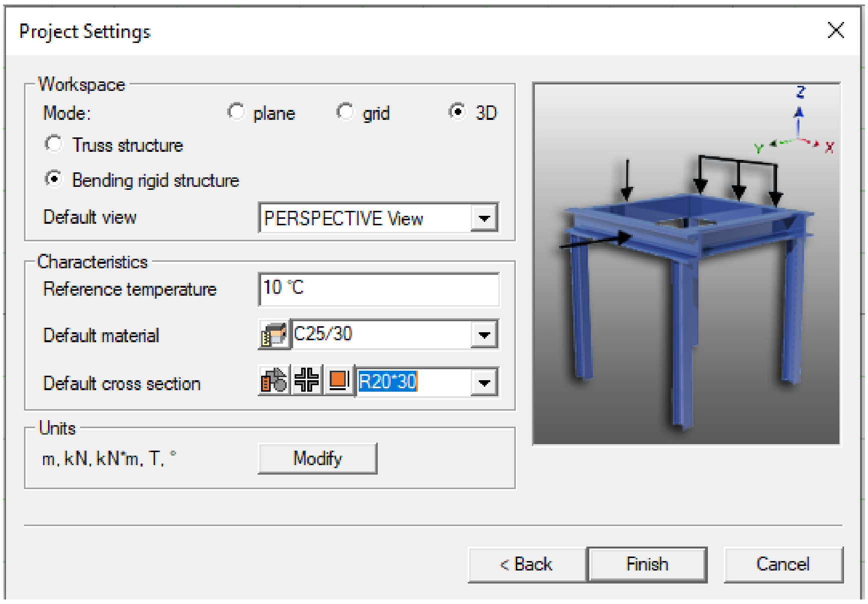


3. Click **OK** and then select **New** from the left side panel to open a new project.
4. Enter details about the current project: name, lot, address, etc. These details are displayed in the reports generated for the current project.



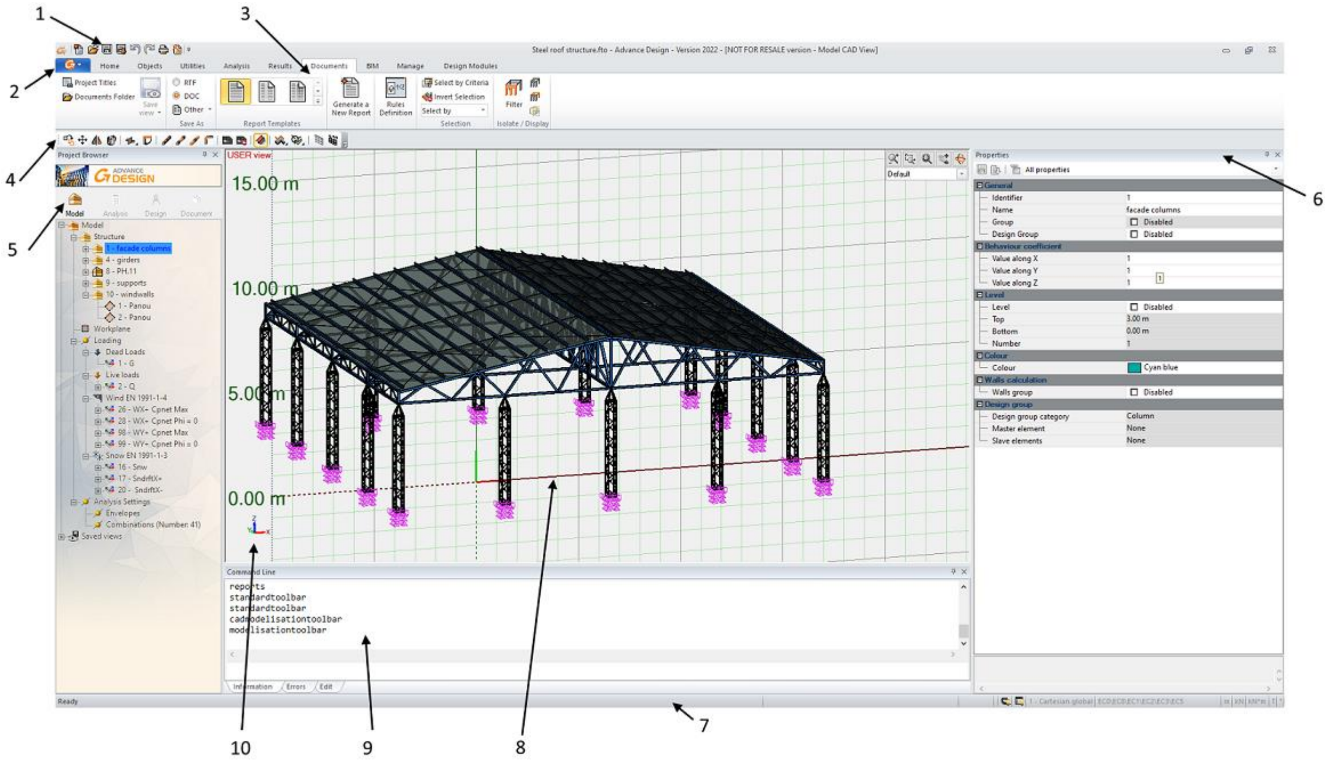
Also, you can open the List of stakeholders dialog box where you can enter details about stakeholders and browse for an image file to add it as a project logo. When the Show in cover sheet option is enabled, the information provided in this dialog box is displayed on the report cover sheet.

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



Advance Design Environment

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



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 objects selection to structure meshing or results post-processing.
6	Properties window	The attributes of all the model entities can be viewed and modified 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.

7	Status bar	The status bar displays information regarding the program status during different phases of the project. It also contains buttons that provide access to the configuration of certain parameters: snap modes, objects tooltips content, current coordinate system, and working units.
8	Graphic area	The graphic area represents the design area of the application; it provides an easy and intuitive use of CAD commands and a realistic rendering of the model. It also allows you to perform practical actions like elements 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 the structure modeling. The workplane's parameters can be defined and the workplane is easily hidden or displayed during the work process.
9	Command line	The command line informs about the status of an action, assists in 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: 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.
10	Coordinate system	The global coordinate system is represented by a three axes 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 an easier manipulation of its different components and commands (i.e., advanced docking, undocking, auto-hiding, tabbing, etc.).

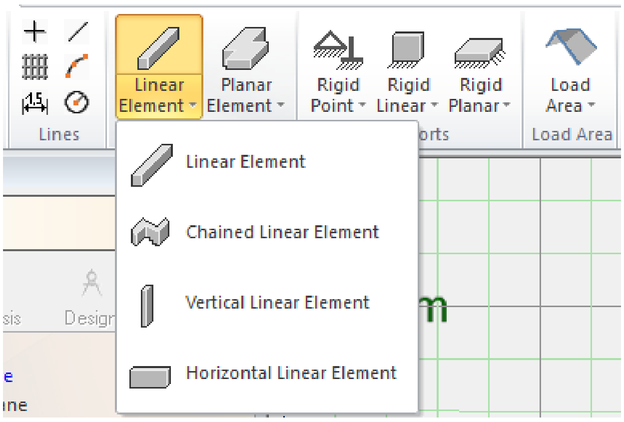
Modeling: Creating the Descriptive Model

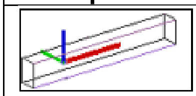
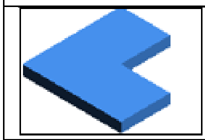
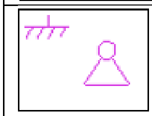
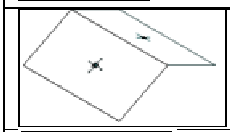
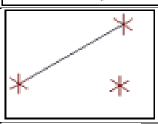
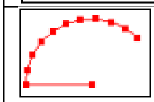
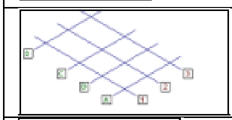
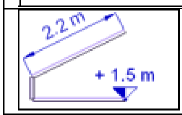
The structure modeling may be done entirely with the use of various CAD tools via the graphic area, where a 2D or 3D representation of the model can be visualized at any time.

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

Advance Design Elements

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



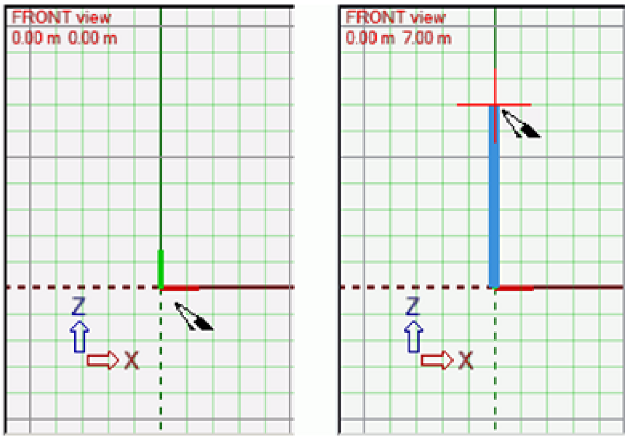
Example	Element type
	Linear elements (bar, beam, short beam, variable beam, tie, strut, cable)
	Planar elements (membrane, plate, shell, plane strain, steel deck)
	Supports (point, linear and planar supports, which can be rigid, elastic or traction only / compression only)
	Load areas: elements used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

Creating elements

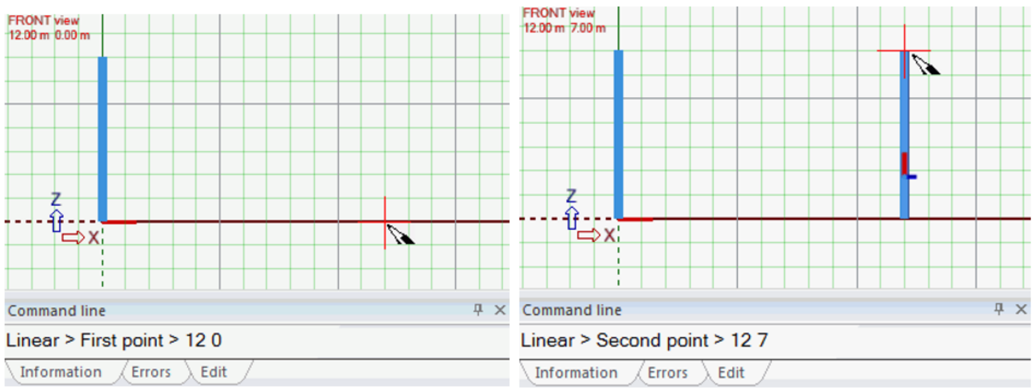
Elements are drawn in the graphic area using the keyboard (by typing coordinates on the command line) or the mouse, relative to the workplane's points or to existing entities. Advance Design also provides various automatic drawing tools (e.g., generate elements on selection, portal frames and vaults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

Example: Creating structure elements

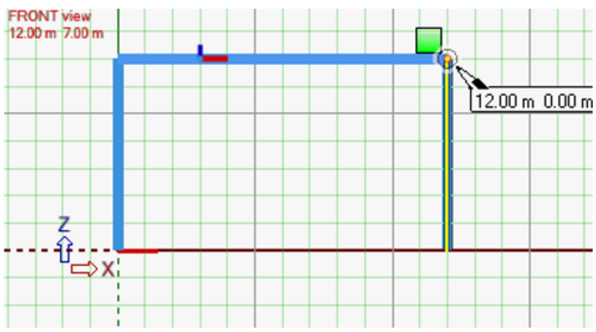
1. On the **Home** tab, **Linear & Planar** panel, select **Linear element.**
2. In the graphic area (XZ plane), click to define a column at **0 0** coordinates for the first extremity and **0 7** for the second one.



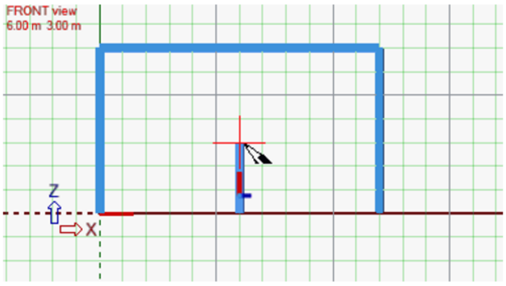
3. With the linear element drawing tool still active, type on the command line the coordinates for the second column: **12 0** for the first extremity and **12 7** for the second one. Input a space between coordinates and press **Enter** after each extremity definition.



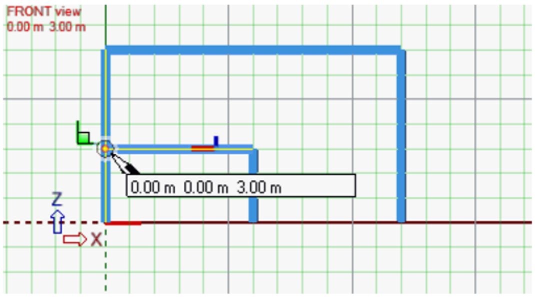
4. Draw the upper beam between the two columns, using the "Extremity" snap mode:



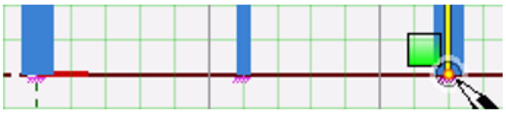
5. Draw a 3 meters high column placed at **6 0** XZ coordinates.



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.

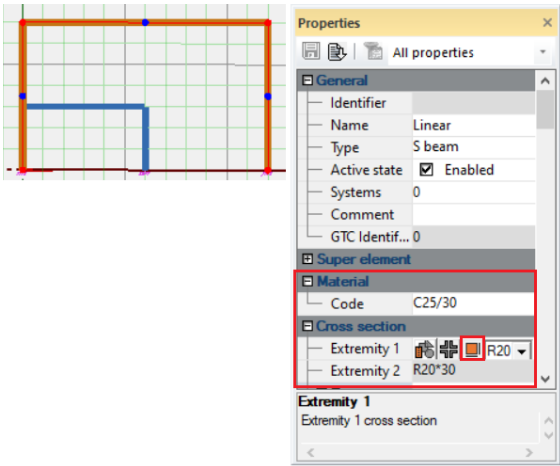


Definition of element properties

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.

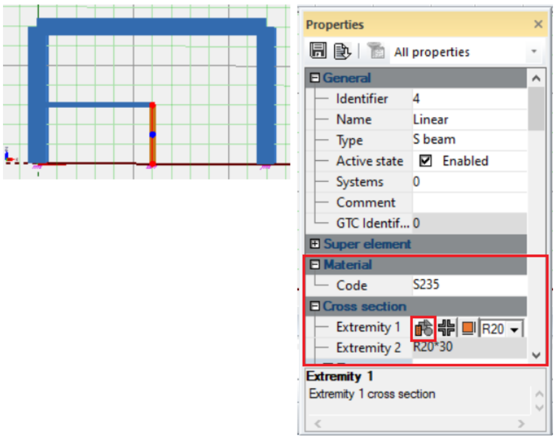
Example: Defining the element properties

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**.

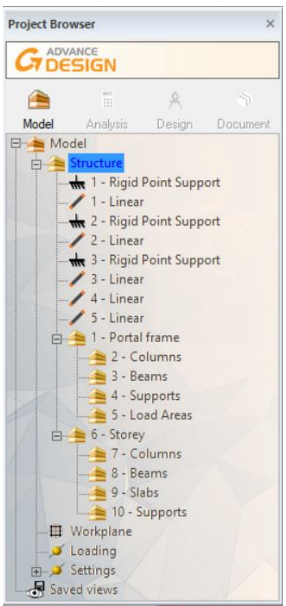
Systems of elements

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 an advanced management of the elements (i.e., hide / display, select, delete, group in systems, etc.).

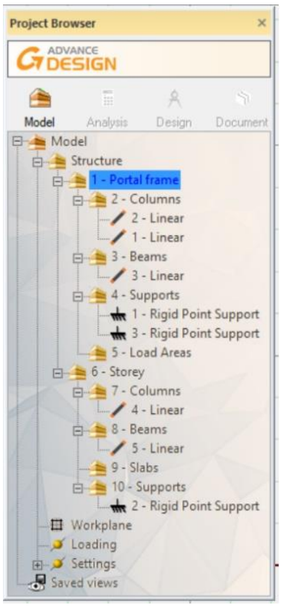
The system concept helps define the behavior of different groups of elements (e.g., assign design templates). Different operations are easily performed on a group of elements using their system's context menu commands. The level function available in the system's properties window defines the level settings 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.



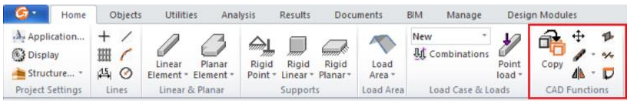
5. In the Project Browser, select the two columns of the portal frame and drag-and-drop them to the **Portal frame > Columns** system.



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

CAD functions

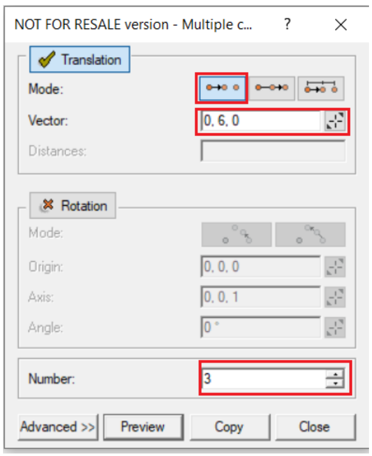
The graphical input of the model is very easy and accurate, using the advanced CAD commands. Easily copy (i.e., by rotation, translation, or symmetry), move, extrude, trim, or extend, subdivide, cut, create openings, etc. using a large set of commands accessible from various locations (i.e., menus, context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



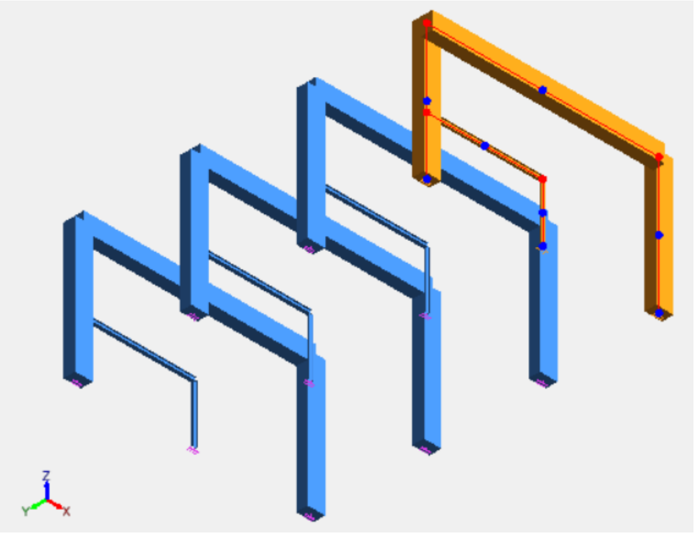
Example: Copying elements

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: **0 6 0**.
   - Number: **3**.

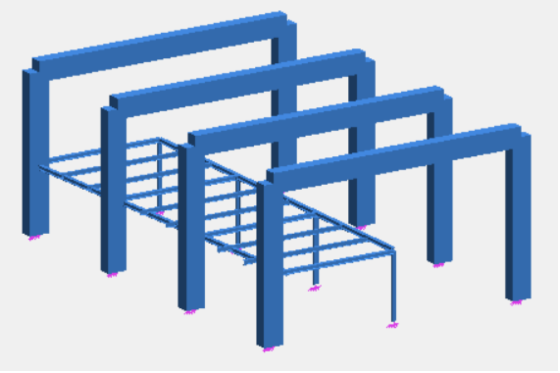


4. Click **Preview** to display the result.
5. Click **Copy** to apply.



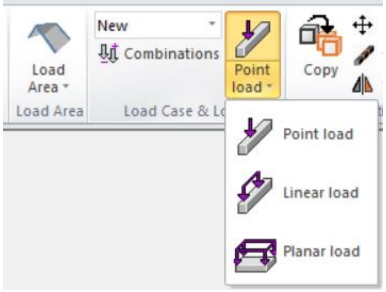
Create the rest of the storey elements:

- 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.



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.

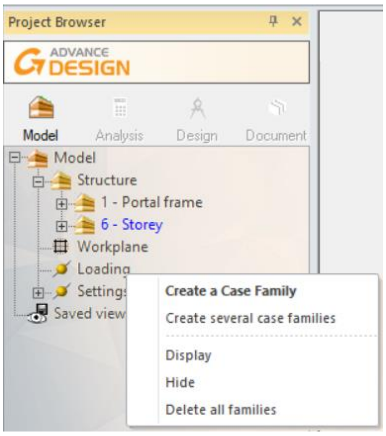


Loads are generated via the graphical input, using the load creation tools available on the **Home** tab - **Load Case & Loads** panel, in the Project Browser or menus. The automatic tools (i.e., pressure loads generator, climatic loads generator, loads on selection, etc.) can also be used.

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).

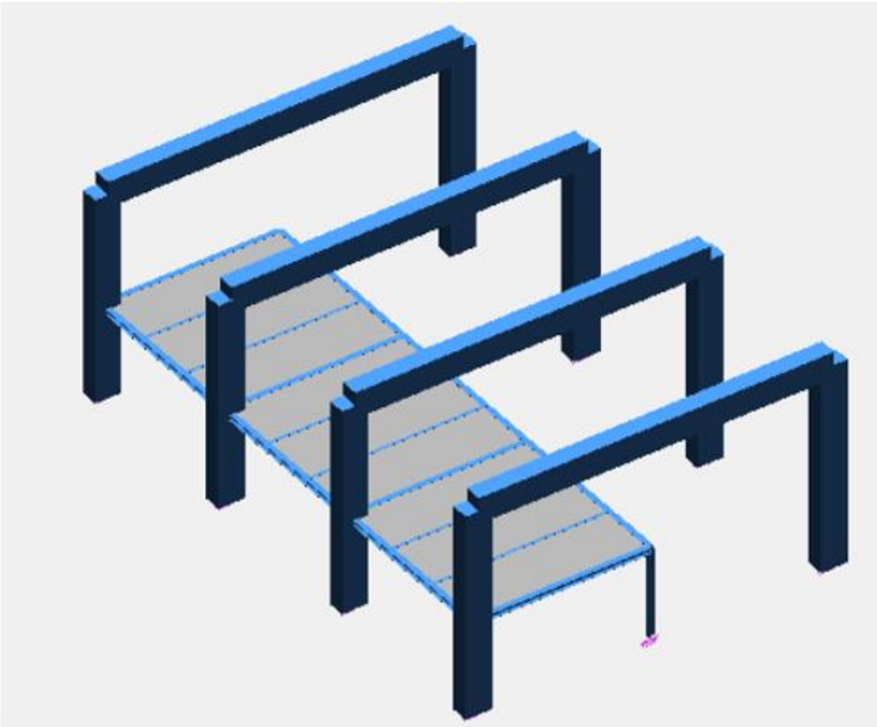
Using the combinations manager, combinations are manually created and the standard combinations available in Advance Design are loaded.



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.



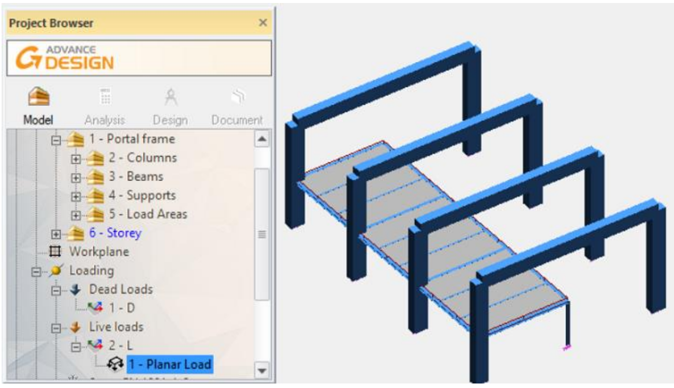
In the slab's properties window define:

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

Global definition

Direction	Type	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

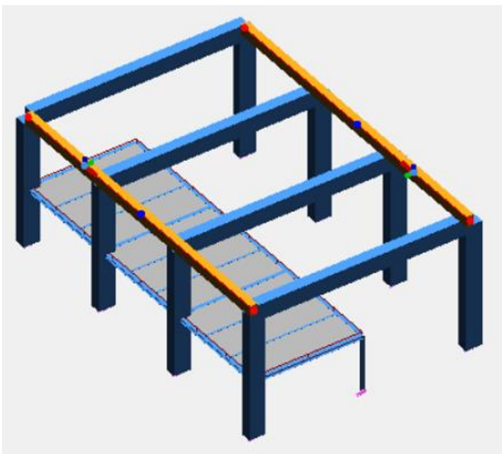
1. In the Project Browser: right click **Loading** and select **Create a case family** from the context menu.
2. In the displayed window, select **Live Loads**.
3. Click **<OK>**. A live loads family and a corresponding case are created in the Project Browser.
4. Select the **2L** live load case in the 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 loads properties window is automatically displayed: input the loads intensity on **FZ: - 5 kN**.
8. Click **<OK>**.



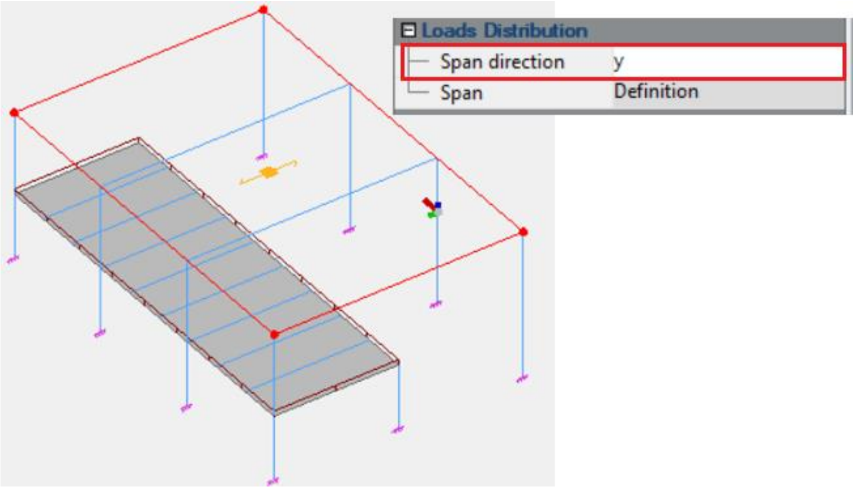
Generating snow loads

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

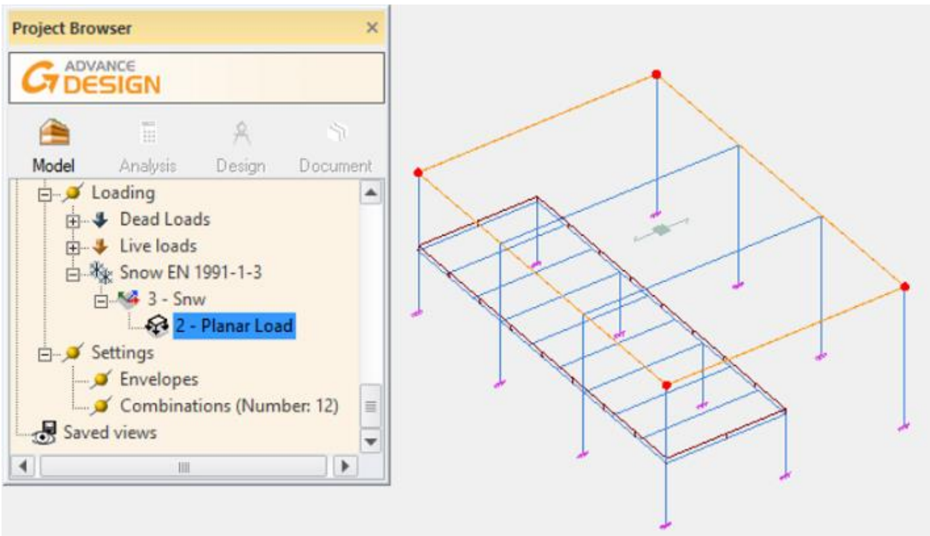
- In the **Portal frame > Beams** subsystem create two longitudinal portal frame beams, using a **C25/30** material and an **R40*60** cross section.



- 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, **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.

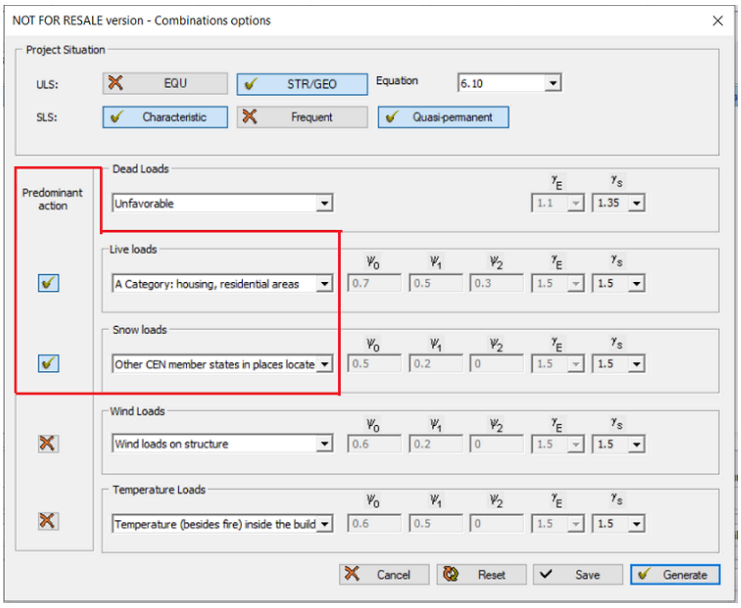


1. In the Project Browser: right click **Loading** and generate a snow family using the same steps described above. A snow family and a snow load case are created in the Project Browser.
2. In the snow family properties window:
   - Enter **0.52** kN/m² in the 'Snow load' field.
3. To automatically generate the snow loads on the load area, select **Generate > Load > Climatic loads** from the menu.



Example: Creating load combinations

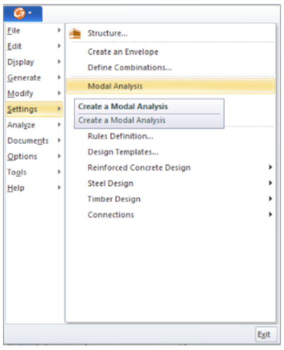
1. In the Project Browser, right click **Combinations** and select **Properties** from the context menu.
2. In the 'Combinations' dialog box, click **Simplified Comb**.
3. Define the live loads and snow loads as predominant actions.
4. Click **Generate**.



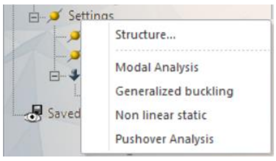
Defining Analyses

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.



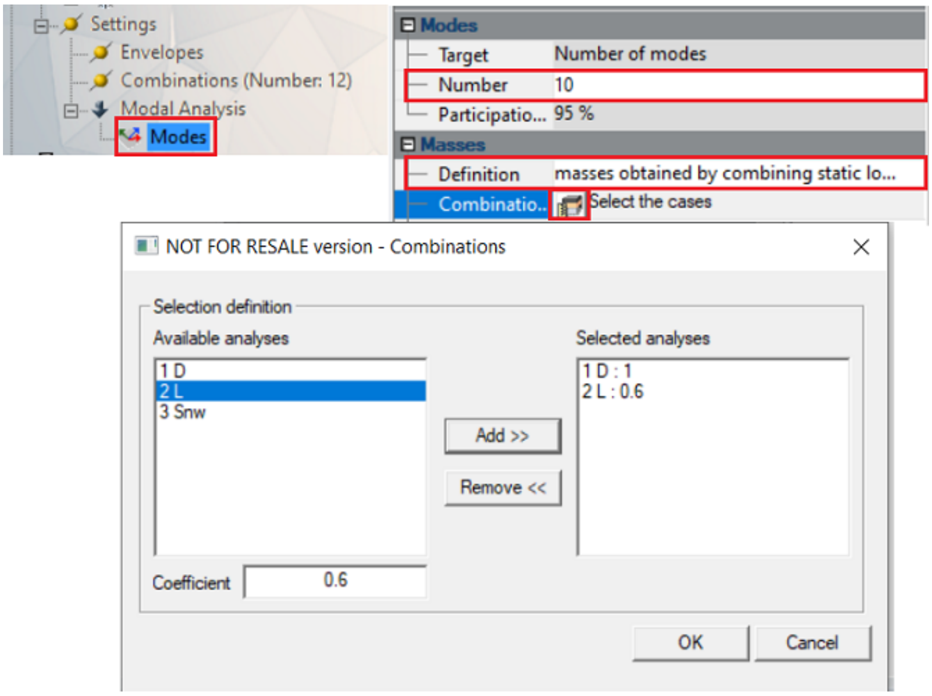
Manage analyses using the Project Browser's commands. View and select the created analyses with the project browser.



The analyses cases parameters are defined in the properties window.

Example: Defining a modal analysis

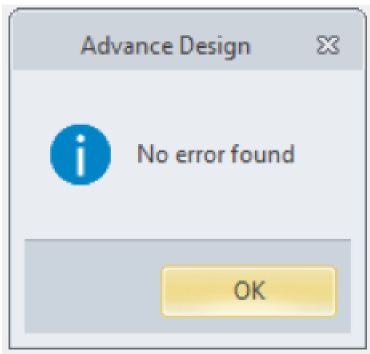
1. Menu: select **Settings > Modal Analysis**.
2. In the Project Browser, a modal analysis family and the 'Modes' case are placed in the 'Settings' group. Select the 'Modes' case to display its properties window.
3. Define the modes' parameters:
   - Number of vibration modes: **10.**
   - Masses definition: select **masses obtained by combining static loads** from the drop-down list and, in the 'Combinations' field below, click  to access the masses combinations dialog box. Define here the following combination: **1*1D + 0.6*2L**.



- Mass percentage on Z: **0%.**

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 (**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.



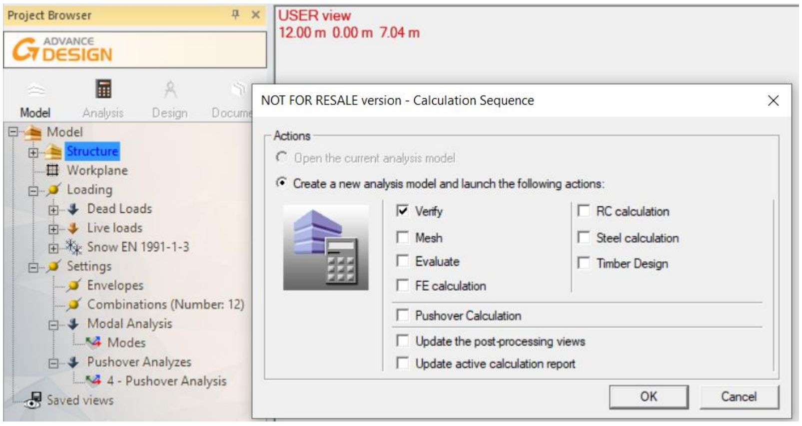
Analysis: Meshing and Calculation

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).

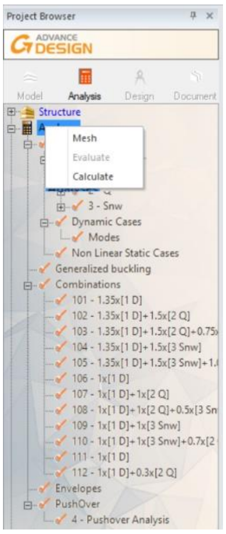
Creating the Analysis Model

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 .

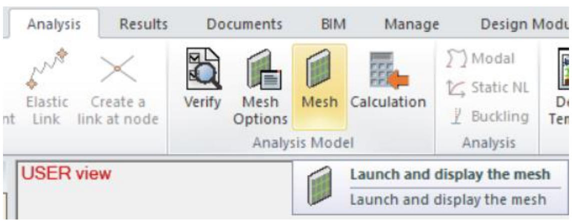
A wizard provides options to combine the desired operations (i.e., verify, mesh, finite elements calculation, reinforced concrete calculation, etc.) into a calculation sequence that is automatically performed.



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.



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.



Meshing

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

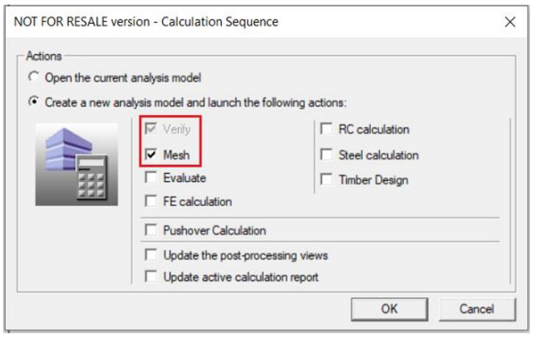
The finite element meshing is performed using the global mesh settings (defined via the **Options > Mesh** command) and the mesh parameters defined for each element (available in the properties window). The mesh parameters of each element are defined using the simplified method (i.e., a meshing density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

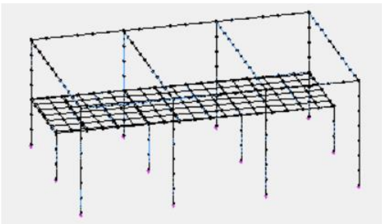
Example: Defining the model meshing

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>**.

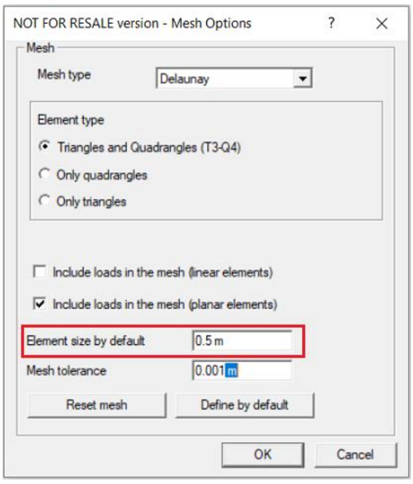
Advance Design creates the analysis model and automatically performs the model meshing.





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>**.



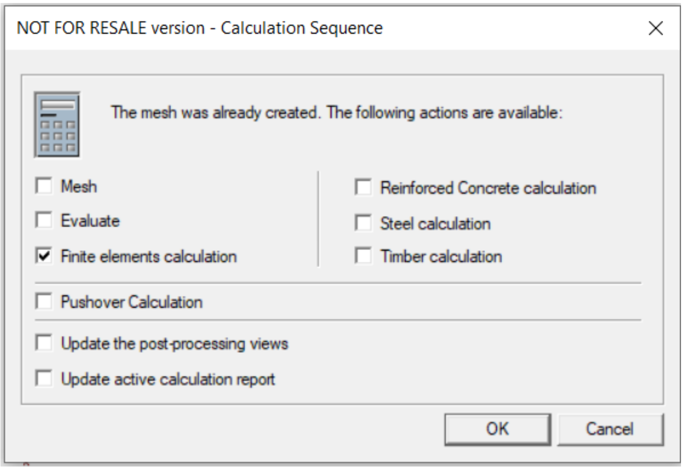
Recreate the mesh

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

The meshing is modified according to the global settings.

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 are selected.

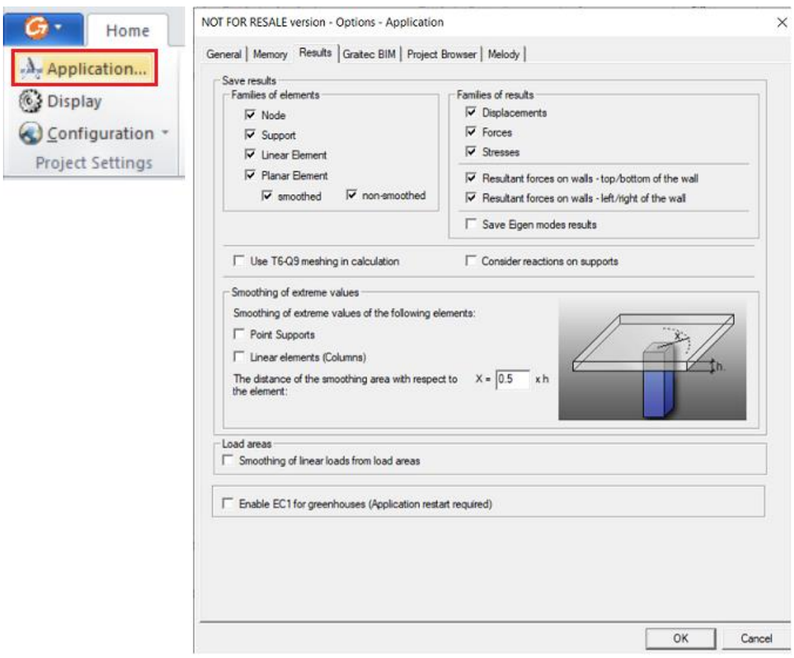


Finite elements calculation

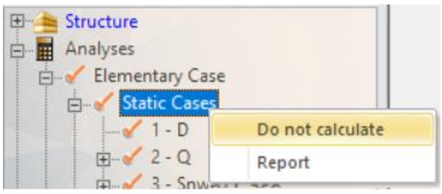
A powerful finite elements engine performs the model calculation according to the structure settings:

- 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).

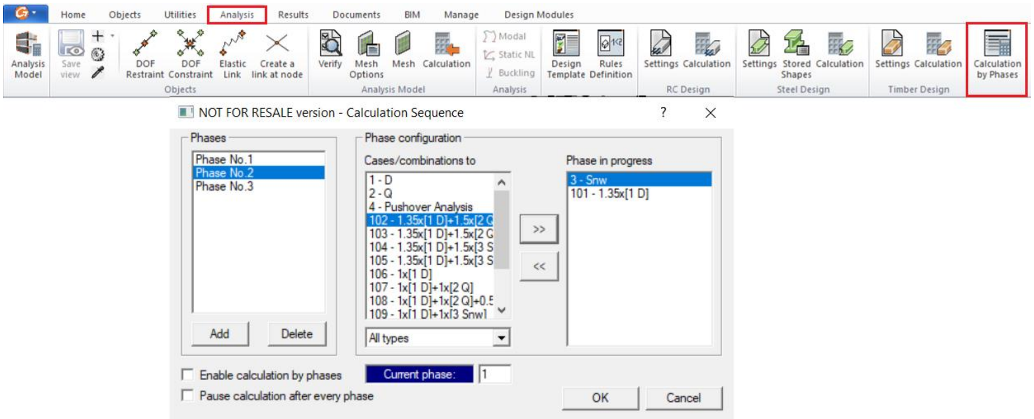
Before the calculation, it is possible to select the elements to calculate and the type of results to obtain, for optimizing the calculation speed and the memory usage.



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



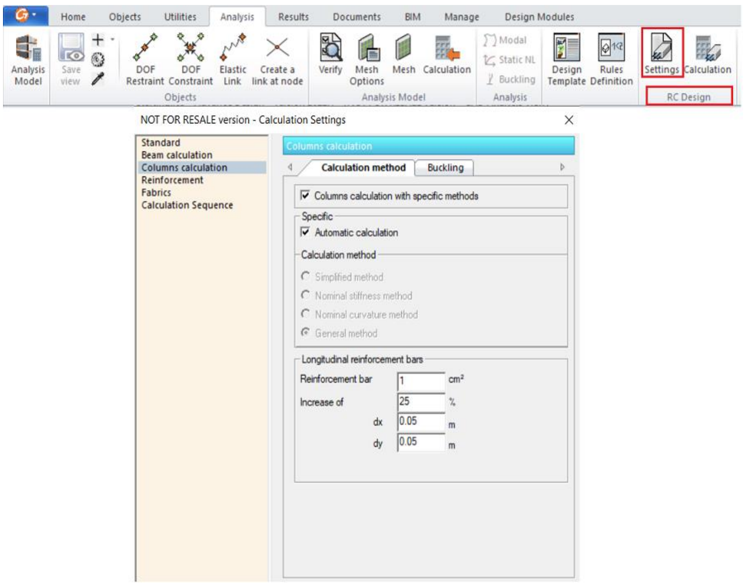
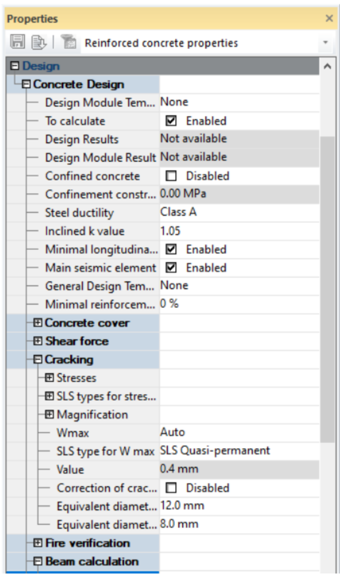
Advance Design can group analyses in calculation phases and calculate them step by step (allowing property modifications for each phase).



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 finite elements calculation was run. The reinforced concrete calculation considers the global and local concrete design settings:

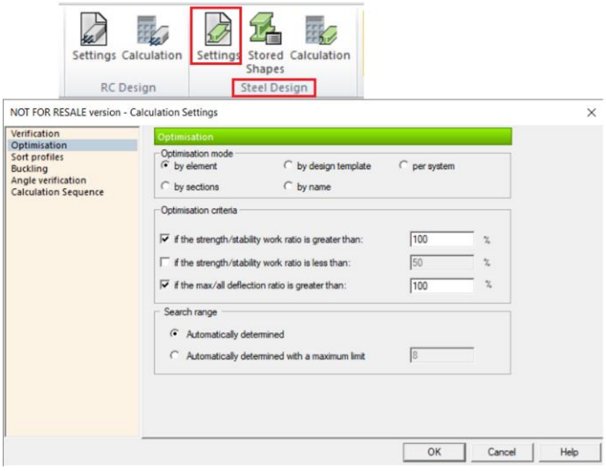
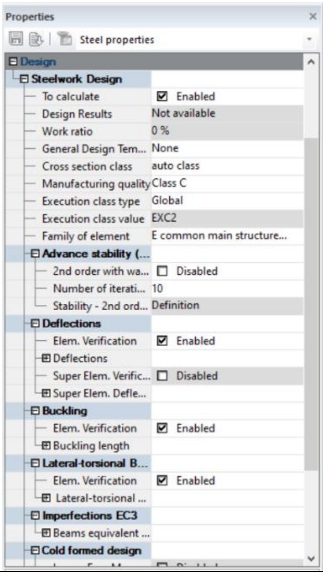
Global concrete settings	*Local concrete settings*
Pertains to the calculation methods of reinforced concrete, the columns verification, reinforcement and buckling parameters, etc.	The local concrete design settings are defined in the properties window of the appropriate elements.
	

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, 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 finite elements calculation was run.

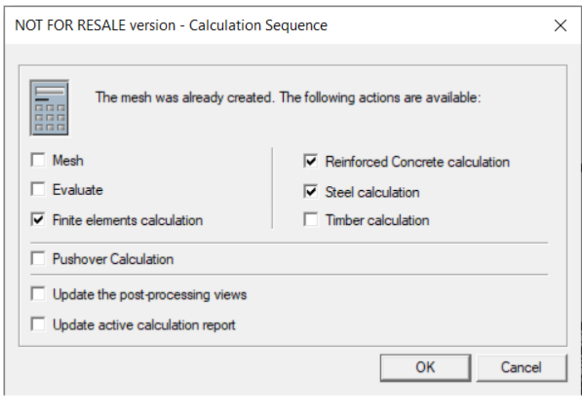
The steel calculation considers the global and local steel design settings:

Global steel settings	*Local steel settings*
Pertains to the steel calculation methods, the optimization criteria, the buckling calculation methods, etc.	The local steel design settings are defined in the properties window of the appropriate elements.
	

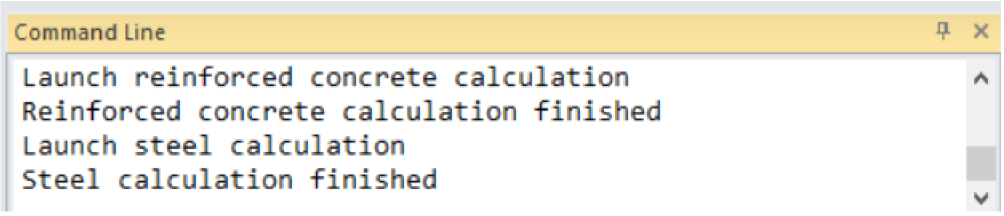
Note: *After the calculation, the results can be viewed and the element parameters modified, if necessary. The desired calculations can be iterated until the appropriate results are obtained.*

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.



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



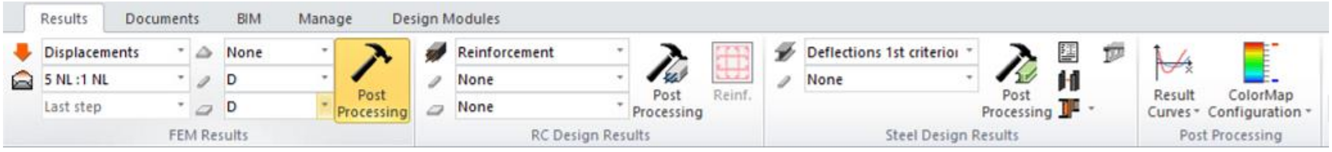
Results Post-processing

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

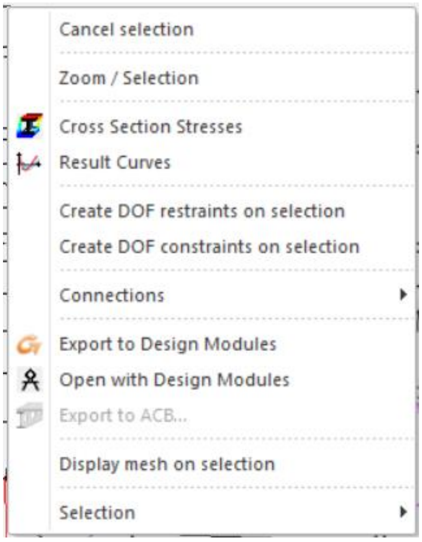
Graphical Visualization of Results

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:

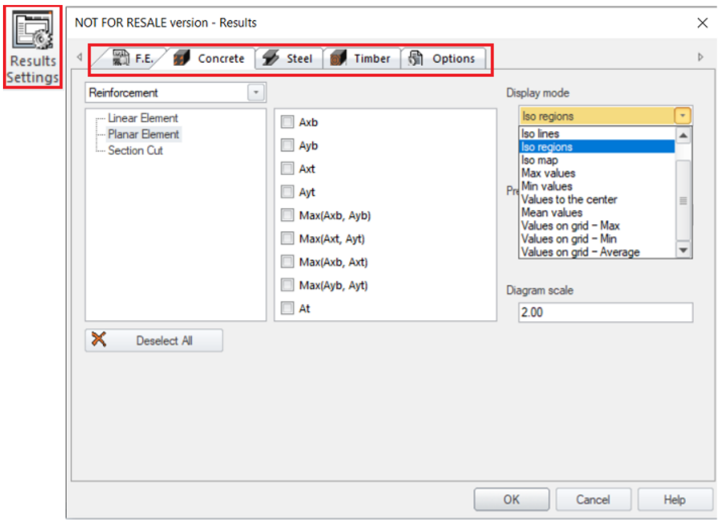
- On the panels in the **Results** tab, which appear automatically once the corresponding calculation is done.



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



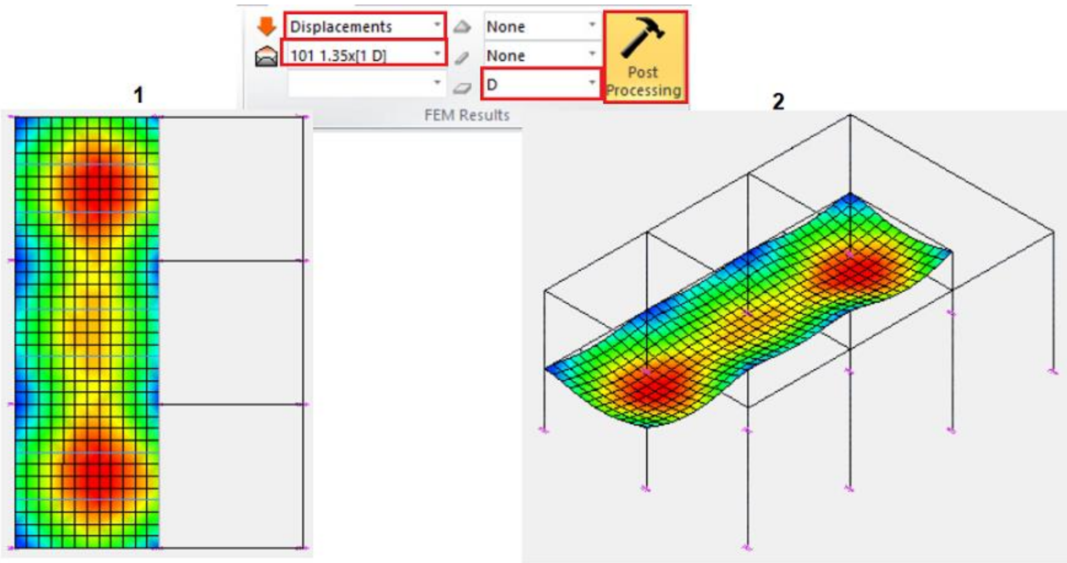
- Using the results configuration dialog box, that provides a detailed configuration of results display. Different visualization modes are available: colors, values, deformed shape, iso-values, iso-regions, vectors, etc.



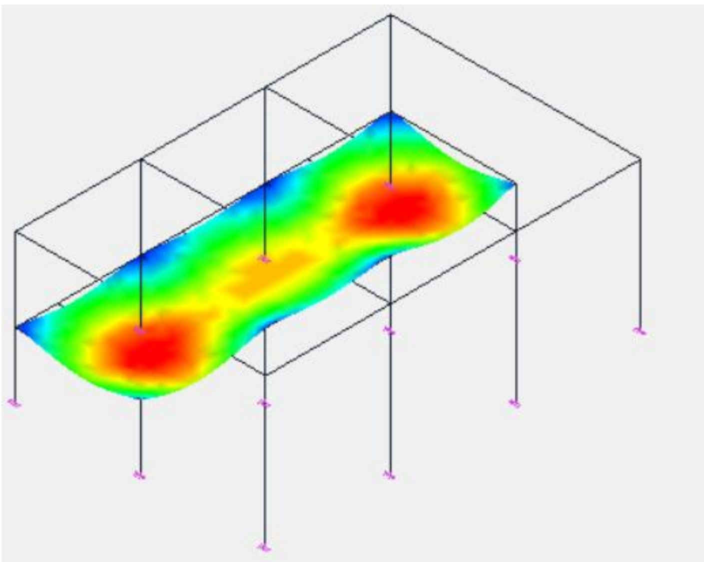
Example: Creating a graphical post-processing of FE results

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 . In the **Result Settings** dialog box - **Options** tab: select **Display results on the deformed** and **Automatic scale of the deformed**.



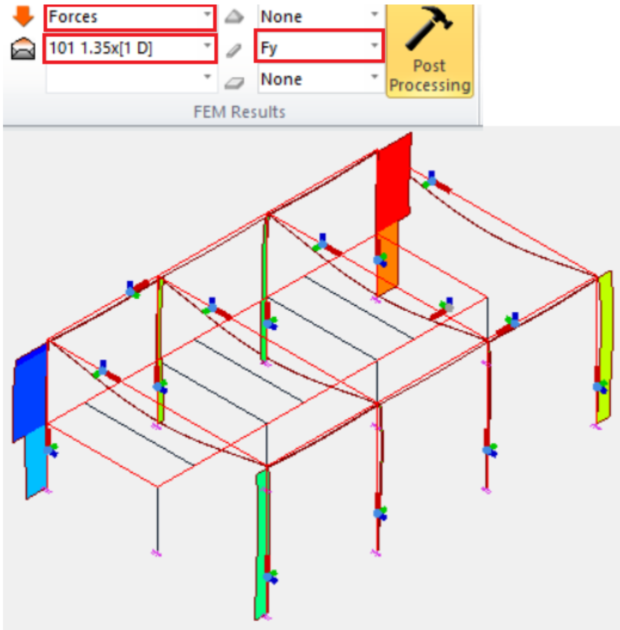
3. Right click the graphic area and deselect **Display the mesh** from the context menu.
4. Open the results configuration dialog box (press **<Alt + Z>**), access the **Options** tab and select **Extreme values**.



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.

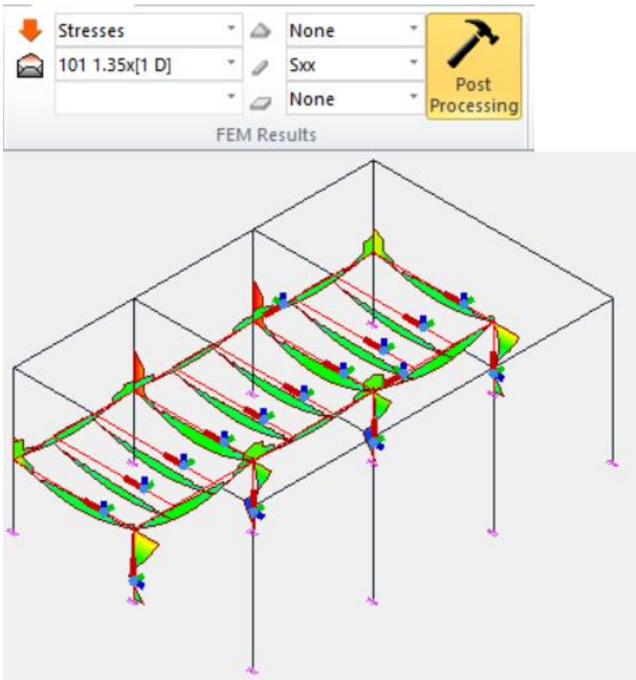
2. On the **FEM Results** panel, select the **Forces** result type, the **Fy** linear elements results and the combination no. **101**. Click **Post Processing** to perform the post-processing.



View 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.



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

Result curves

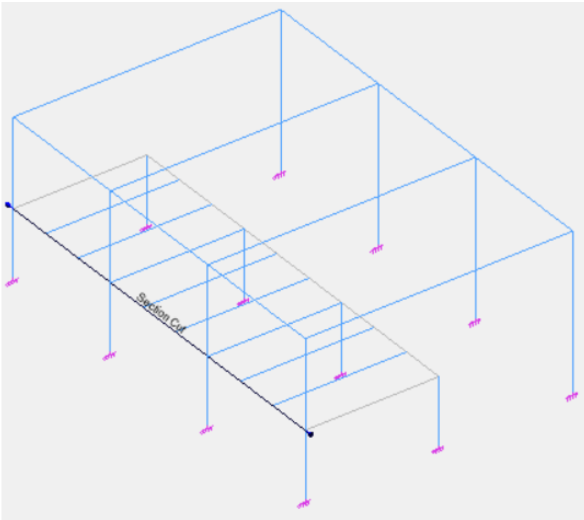
Different results (i.e., FE results as displacements, forces, stresses, and concrete reinforcement area) can be viewed using the "Result curves" command available in the post-processing step of the project.

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

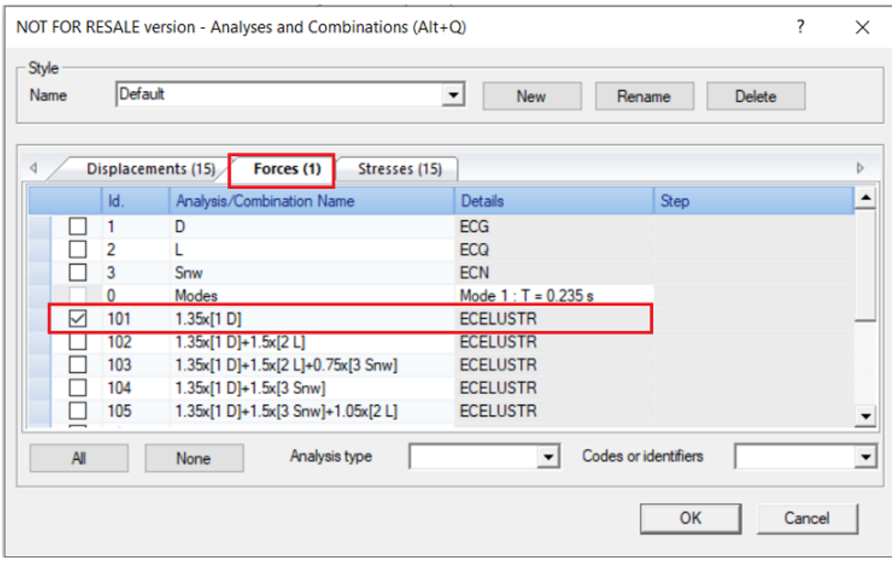
First, create a section cut:

- 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.

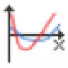


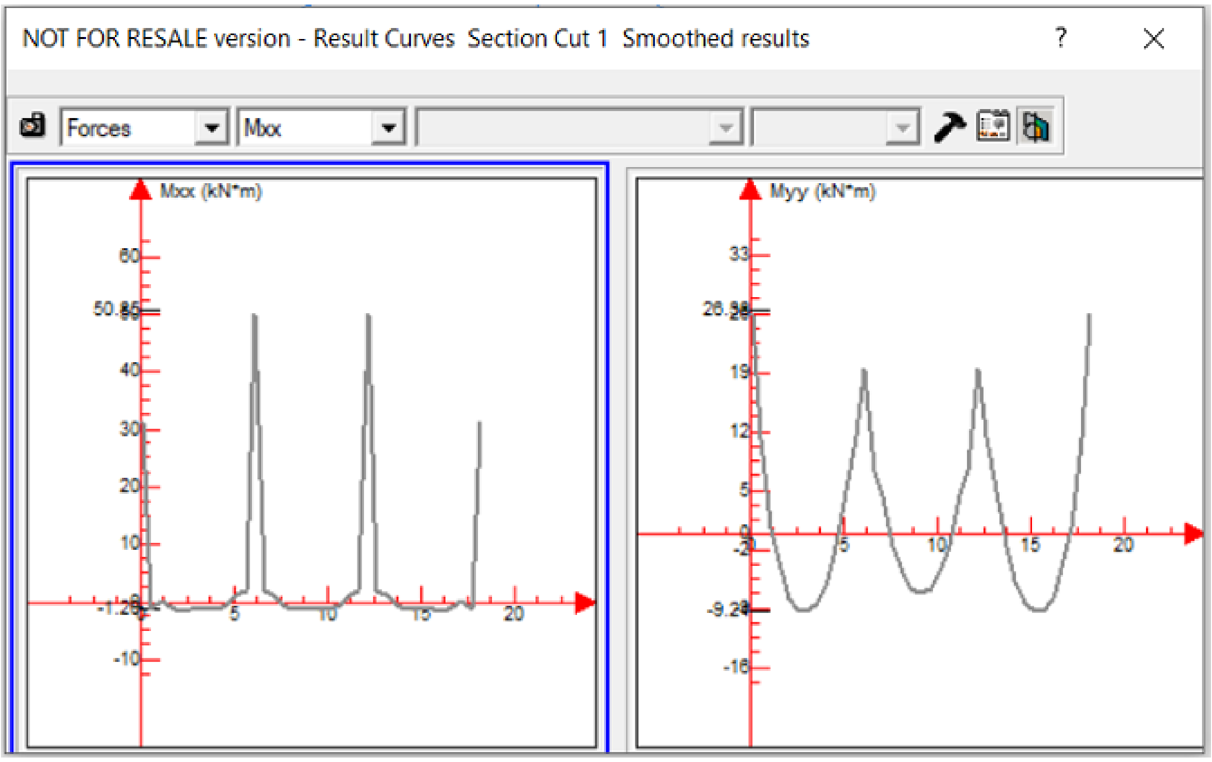
Then, select the analyses displayed on the curve and view the results:

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



3. Select the section cut.

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

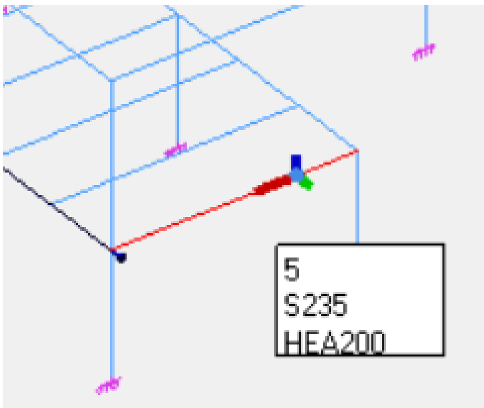


Stresses diagrams

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.

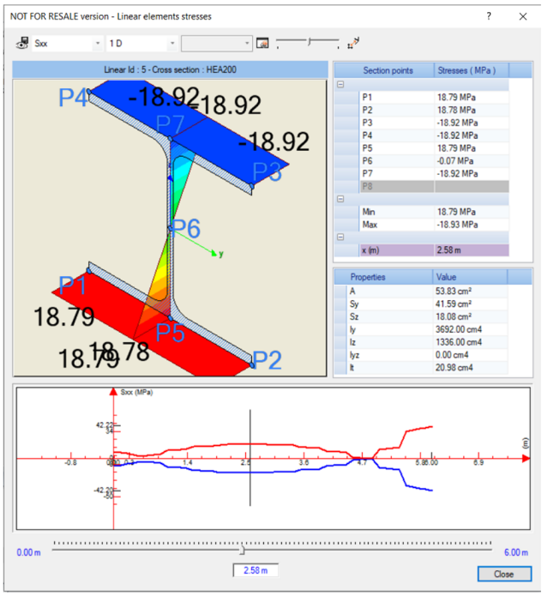
Example: Displaying a stresses diagram

1. 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 the beam of interest (i.e., **S235** material, **HEA200** cross section) - click to select it.



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

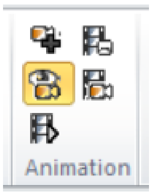
The sections stresses diagram is displayed in a new window. Use the slider to view the stresses on each point on the beam's length.



Post-processing animation

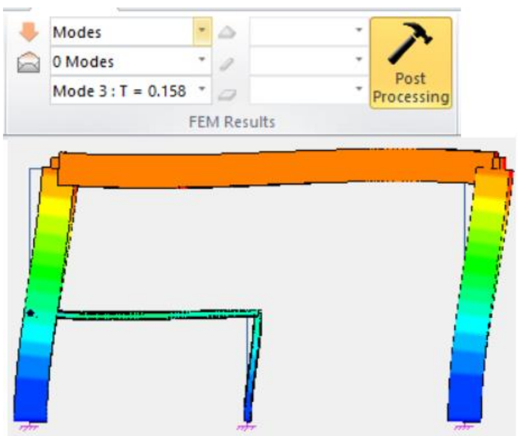
An animation can be created in Advance Design, starting from the graphical results post-processing, following the results distribution and the deformed shape of the structure.

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



Example: Creating a post-processing animation

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



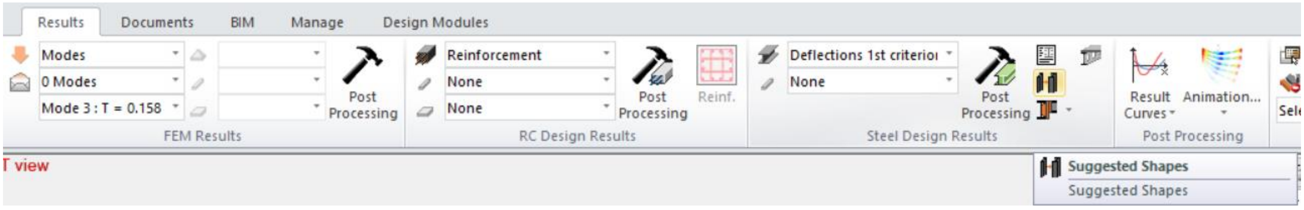
3. On the **Results** tab, **Post Processing** panel, open the **ColorMap Configuration** drop-down list and select **Animation** to view the post-processing results in animation.

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

Design Post-processing

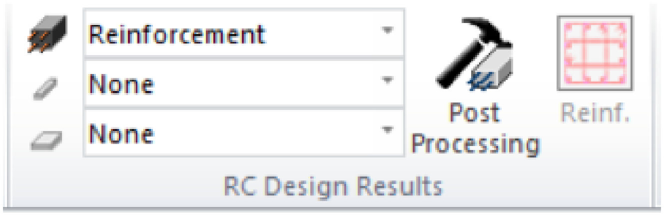
In the post-processing phase, once the corresponding calculations are performed, the results of the concrete / steel verifications can be viewed. Moreover, the concrete and steel members of the structure can be optimized using the functions provided by these design modules.

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



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.

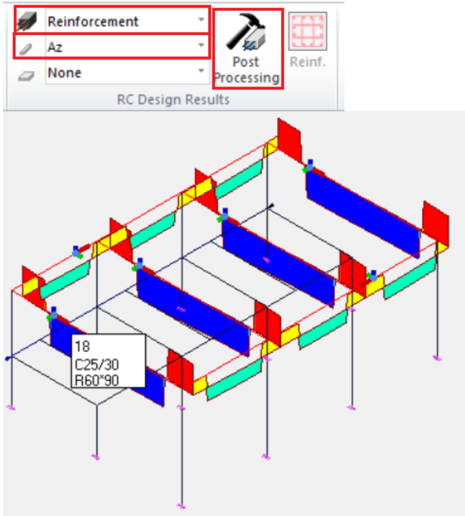


In the properties window of certain concrete elements (columns), the interaction curves issued from the reinforcement parameters, which are either determined automatically by the concrete module or set by the user, can be viewed. Therefore, it is possible to adjust, for example, a highly slender column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

Example: Viewing the longitudinal reinforcement on beams

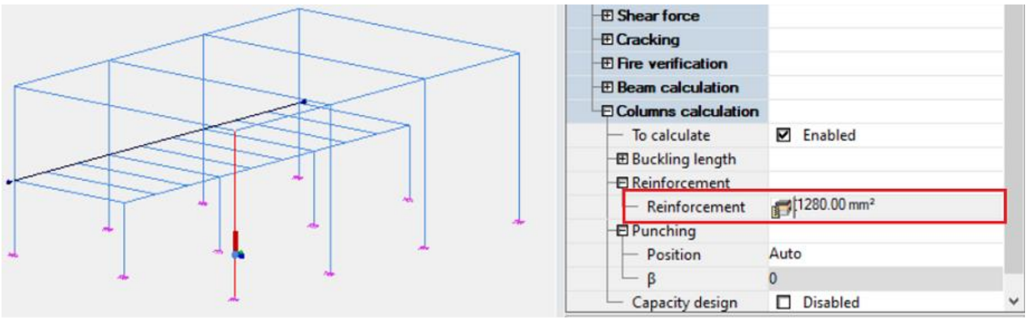
1. Define a (-1, -1, 1) view of the workplane by pressing **<Alt + 6>.**
2. In the Project Browser, select **Portal frame > Beams** system and press **<Space>**.
3. On the **Results** tab, **RC Design Results** panel:
   - Select the result type: **Reinforcement.**
   - Select the result on linear elements: **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.

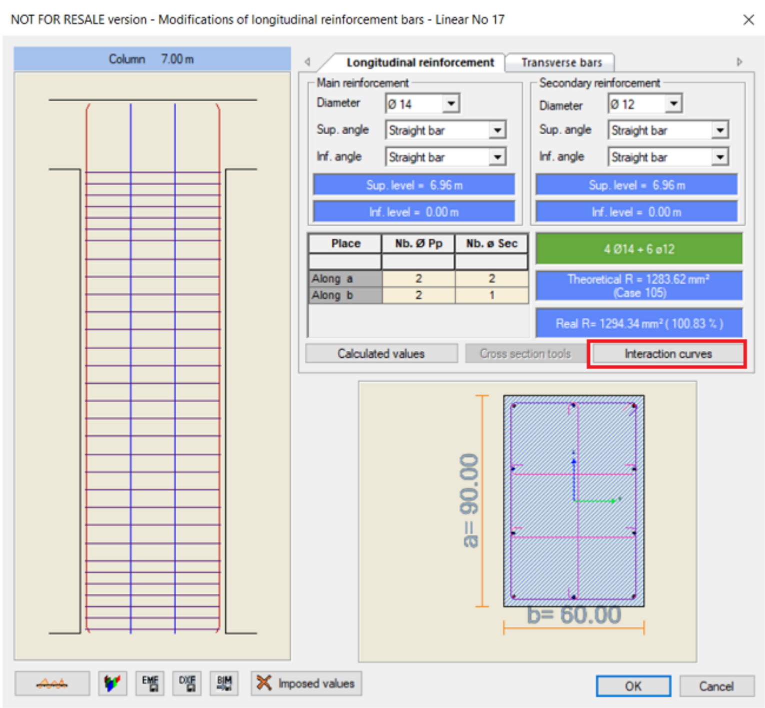


Example: Viewing the reinforcement results on a column

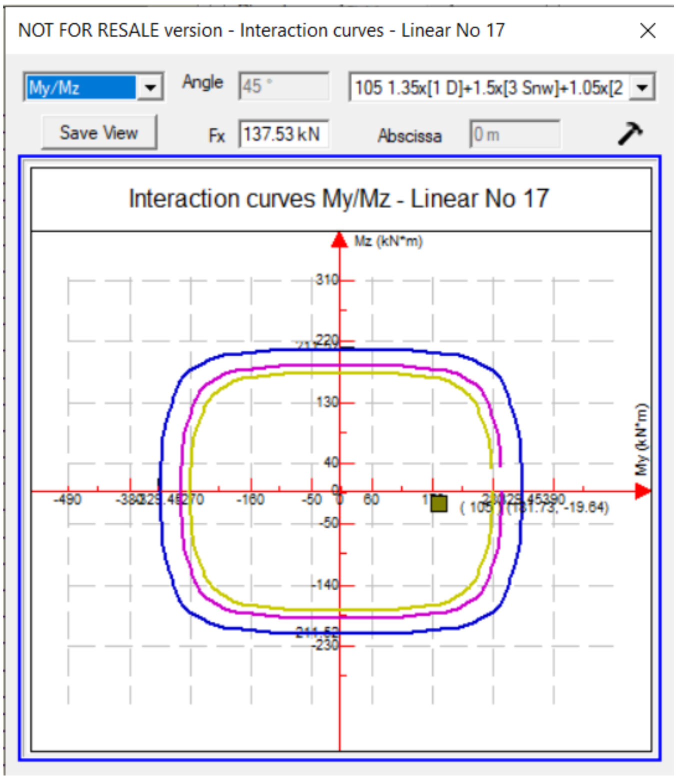
1. Select one column of the portal frame.
2. In the element properties window, go to the **Design** category > **Reinforcement** field.



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



4. Click **Interaction curves** to access the interaction curves window.

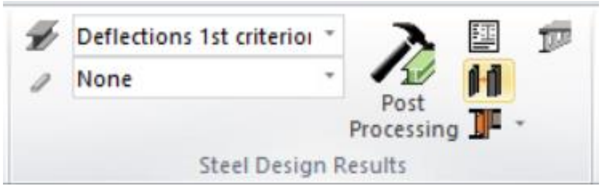


It is possible to view the position of the force component relative to the interaction area. For advanced visualization options, double click the diagram and the interaction curve is displayed in a new window.

Steel results

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

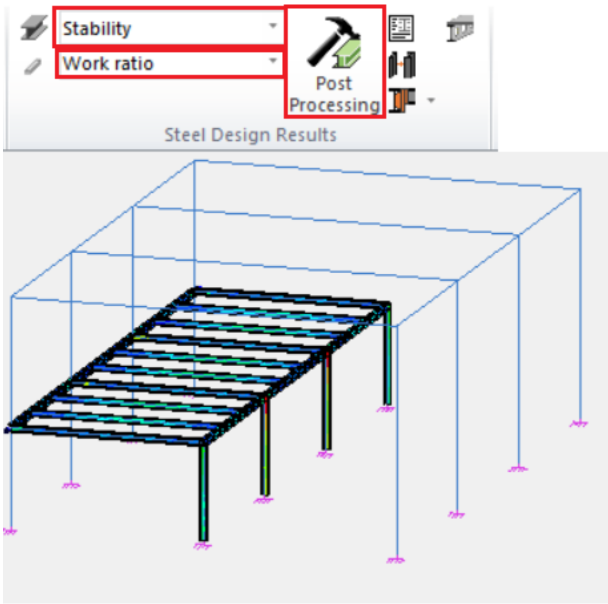
The steel results post-processing commands are available on the Results tab, Steel Design Results panel once the steel calculation is done.



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.**

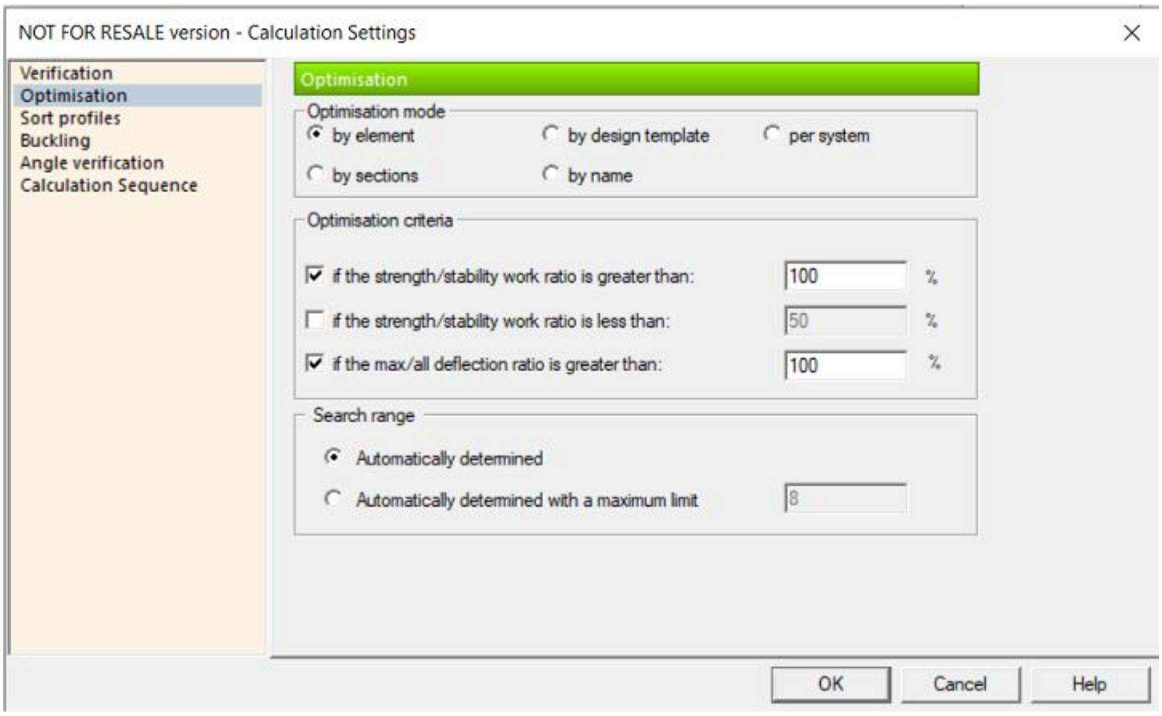


2. Access the 'Results' dialog box by pressing **<Alt + Z>**.
3. Access the **Options** tab and select **Extreme values**.
4. Click **<OK>**.

Steel elements optimization

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

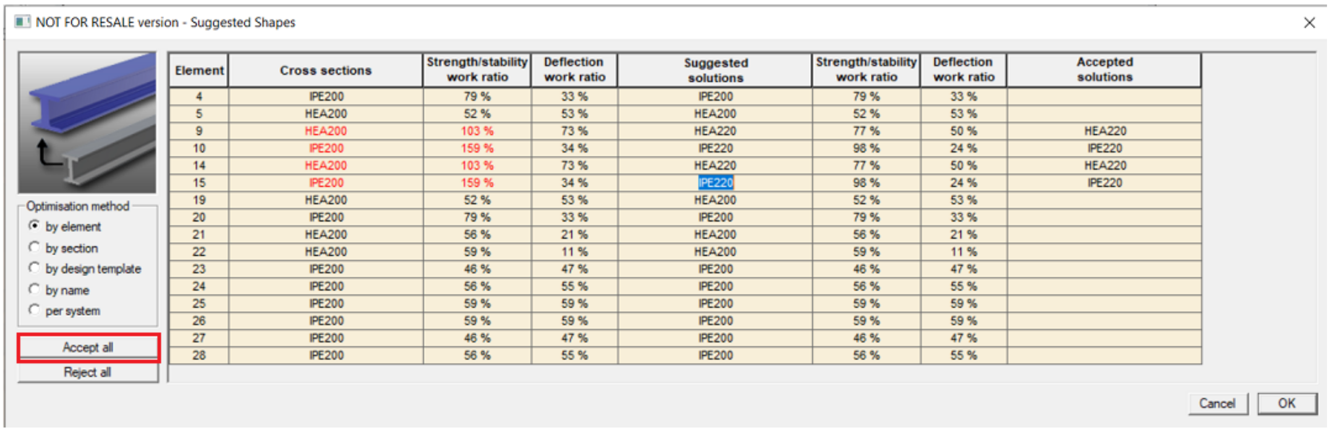
The program identifies the steel shapes with a higher / lower work ratio than specified and suggests more appropriate cross sections.



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 . 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	IPE200	79 %	33 %	IPE200	79 %	33 %	
5	HEA200	52 %	53 %	HEA200	52 %	53 %	
9	HEA200	103 %	73 %	HEA220	77 %	50 %	HEA220
10	IPE200	159 %	34 %	IPE220	98 %	24 %	IPE220
14	HEA200	103 %	73 %	HEA220	77 %	50 %	HEA220
15	IPE200	159 %	34 %	IPE220	98 %	24 %	IPE220
19	HEA200	52 %	53 %	HEA200	52 %	53 %	
20	IPE200	79 %	33 %	IPE200	79 %	33 %	
21	HEA200	56 %	21 %	HEA200	56 %	21 %	
22	HEA200	59 %	11 %	HEA200	59 %	11 %	
23	IPE200	46 %	47 %	IPE200	46 %	47 %	
24	IPE200	56 %	55 %	IPE200	56 %	55 %	
25	IPE200	59 %	59 %	IPE200	59 %	59 %	
26	IPE200	59 %	59 %	IPE200	59 %	59 %	
27	IPE200	46 %	47 %	IPE200	46 %	47 %	
28	IPE200	56 %	55 %	IPE200	56 %	55 %	

2. Click **Accept all** to accept the suggested shapes.
3. Click **<OK>** to close and apply.
4. Rerun the finite elements 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 %

Saved post-processing views

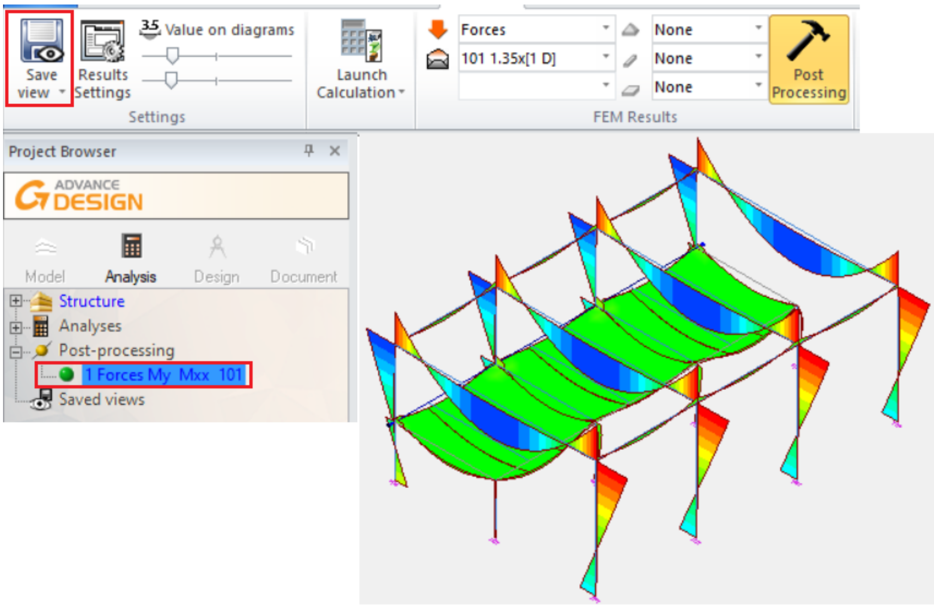
A post-processing view saves the entire post-processing scenario (i.e., result type, result component, selected analyses and elements, results visualization settings) together with the display settings of the model (i.e., viewpoint, rendering, etc.). For each post-processing view, a corresponding image 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.



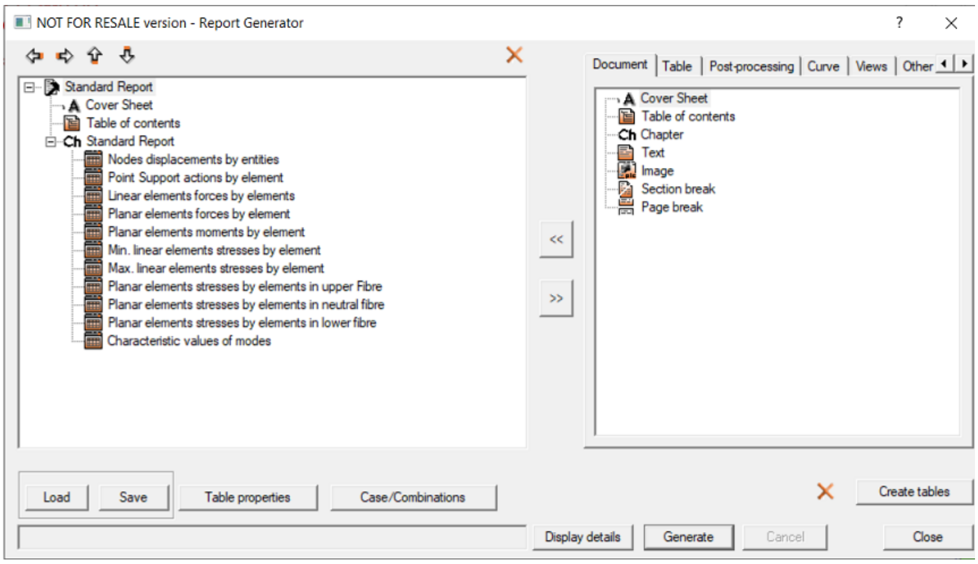
To access a saved post-processing view: double click it in the Project Browser.

Reports

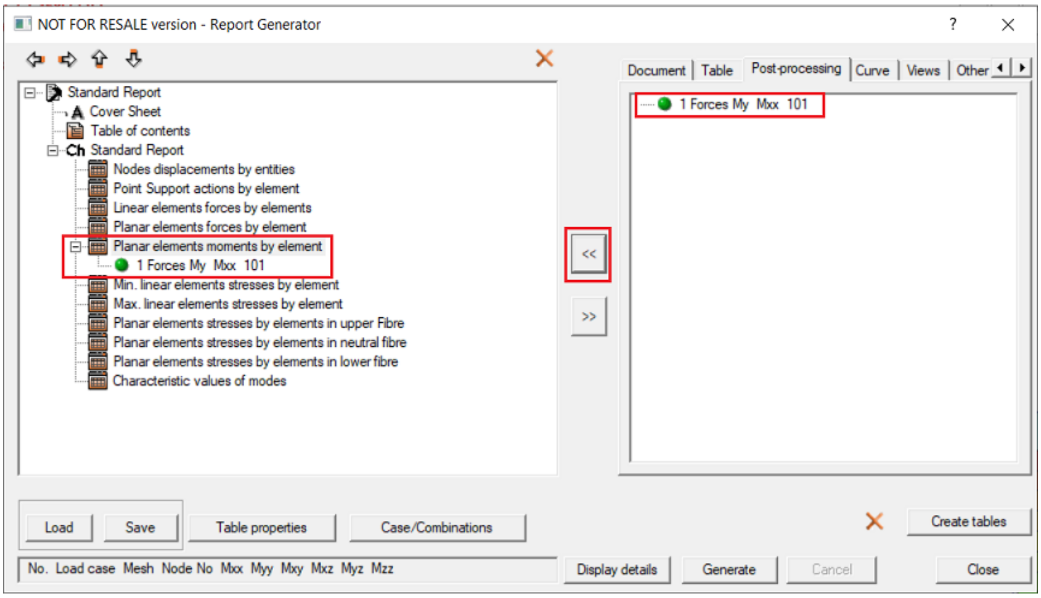
Advance Design provides an advanced and powerful report generator tool, with which the desired reports are easily defined. The available report templates can be used, or new templates can be defined. The report generator filters its content according to the current settings and available results. At 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.



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.



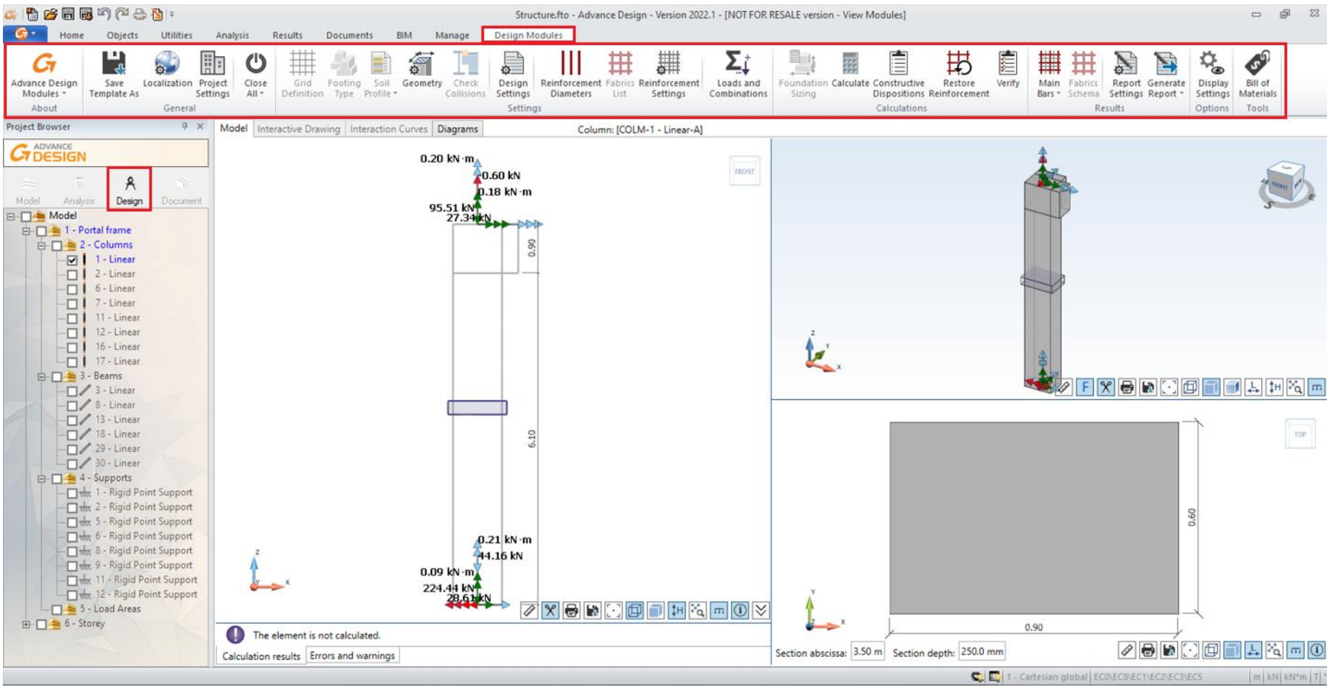
4. Click **Case / Combinations** and select only combination **101** in the **Displacements, Forces** and **Stresses** tabs.
5. Click **Generate** to start the report creation. When finished, the report is automatically displayed with the document viewer application.

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 an analysis is performed, and an element is selected in the Design tab of the Project Browser by double clicking on it.

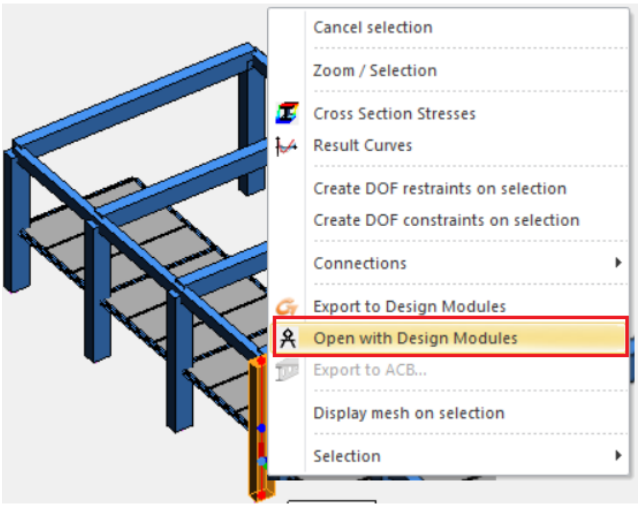
The Design Modules tab of the Advance Design Ribbon provides you with access to commands from Reinforced Concrete module and Steel Connection module. This way, you can achieve the entire design process within Advance Design.



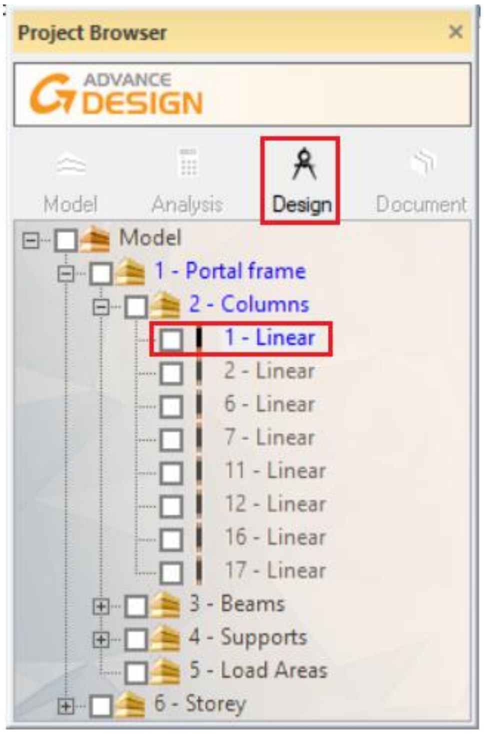
Concrete Elements

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

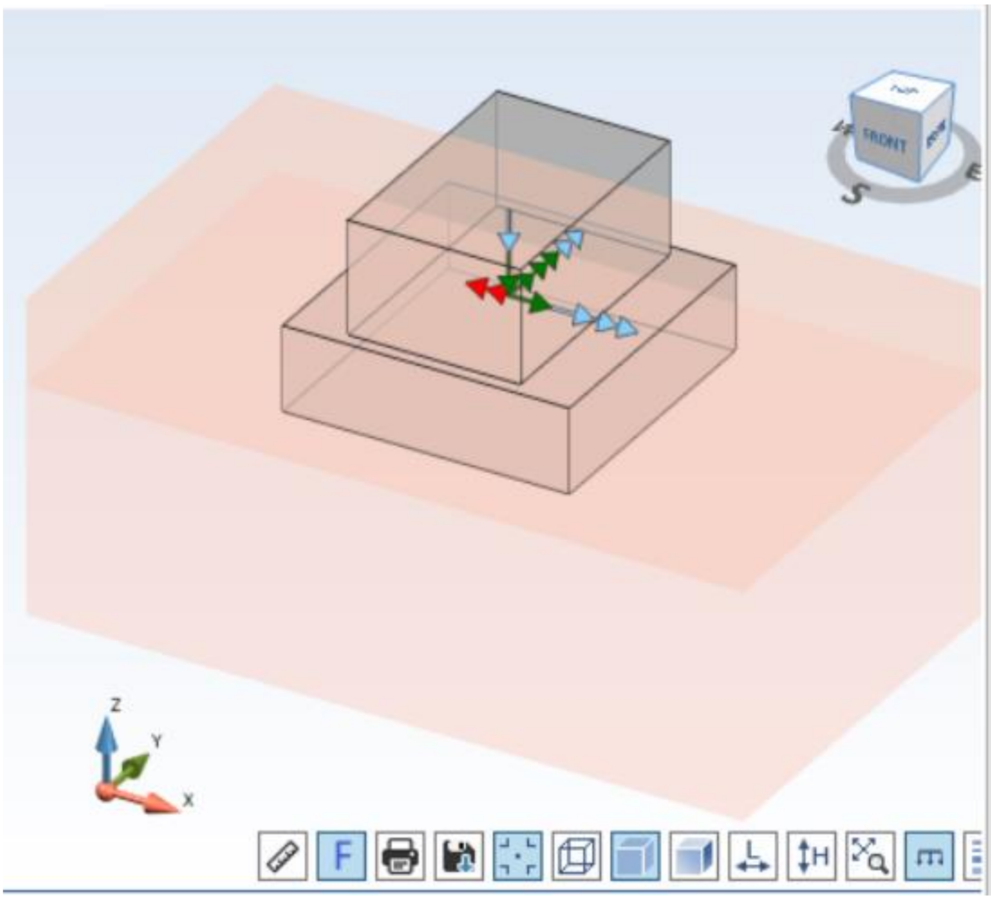
- applying the **Open with Design Modules** command from the contextual menu:

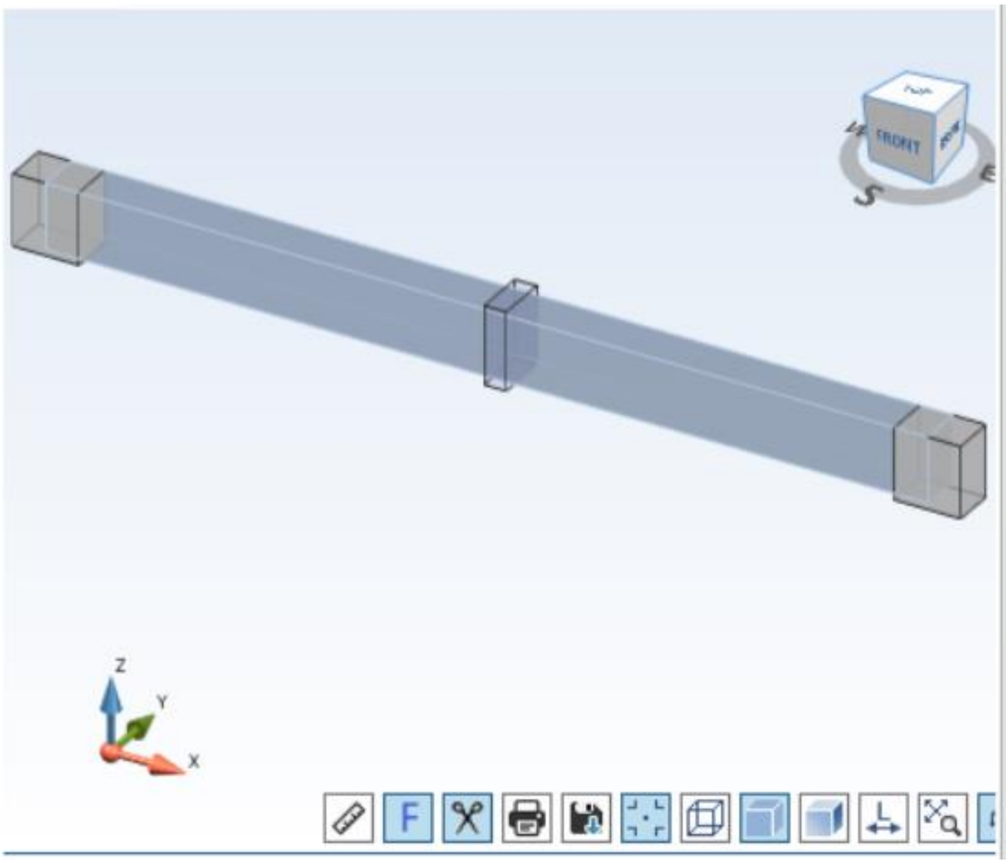


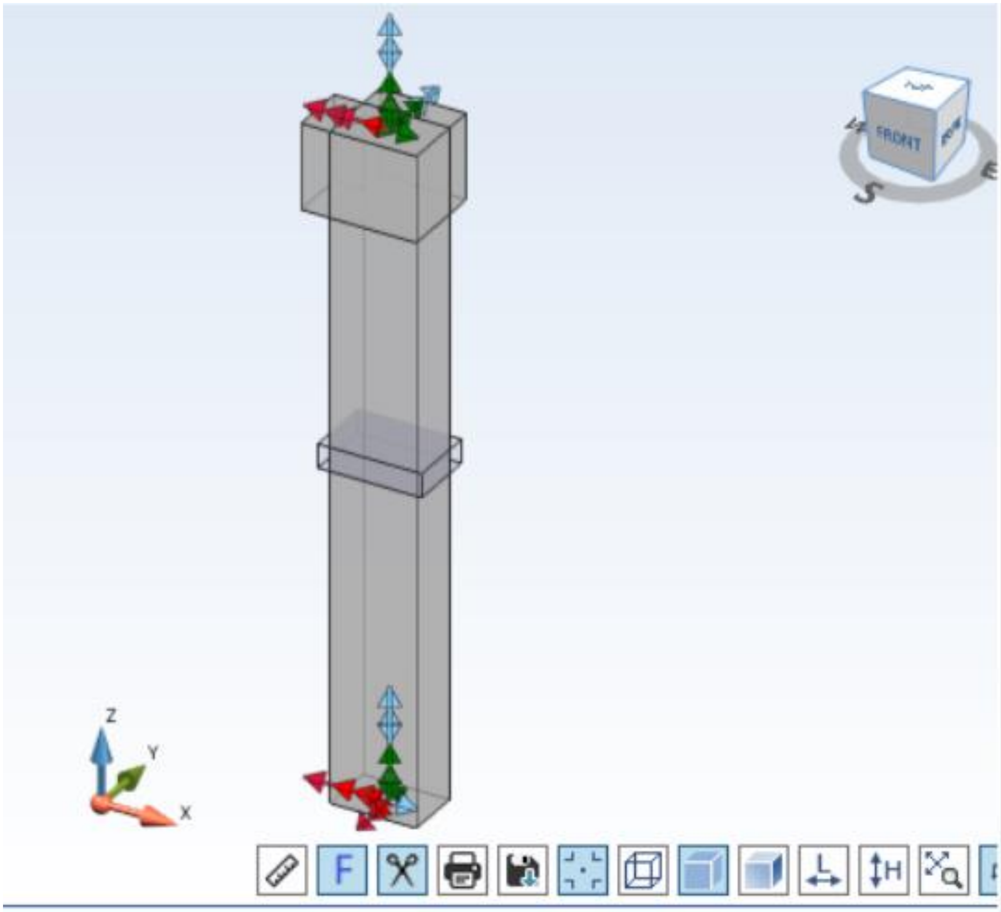
- by double clicking on an element in the **Design** tab of the Project Browser:

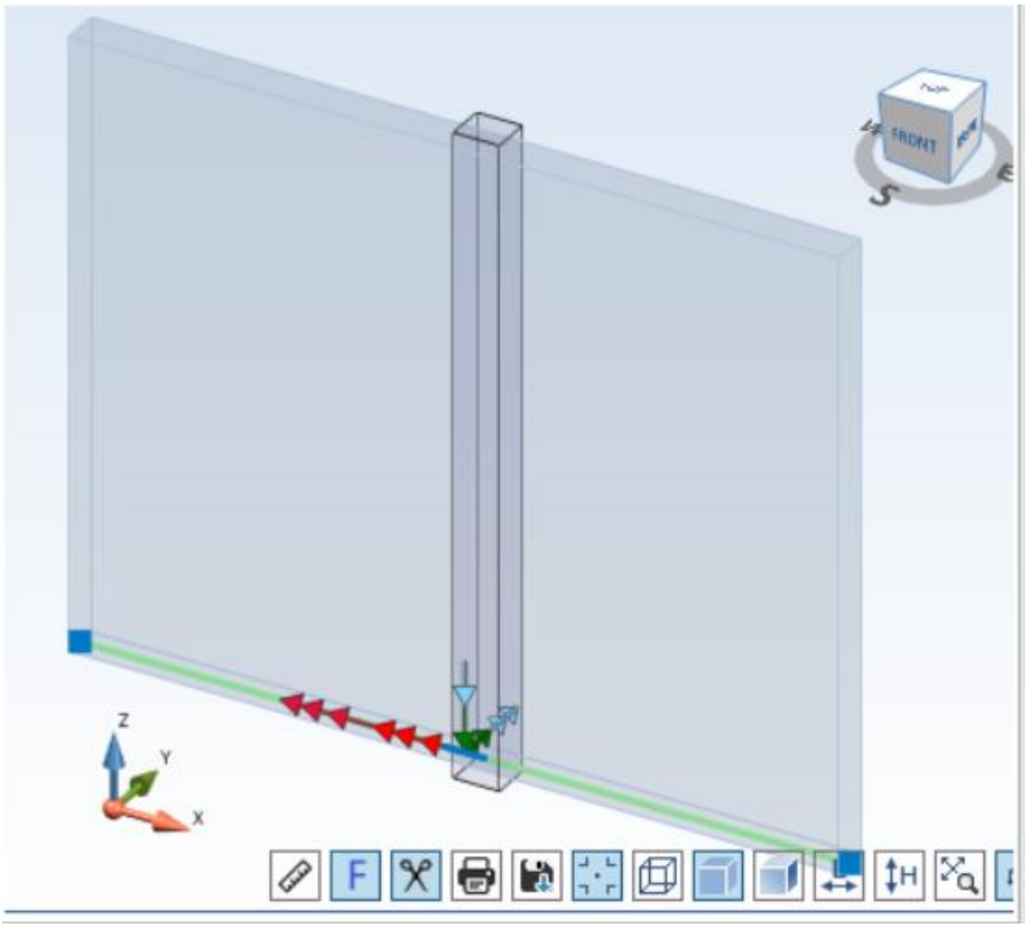


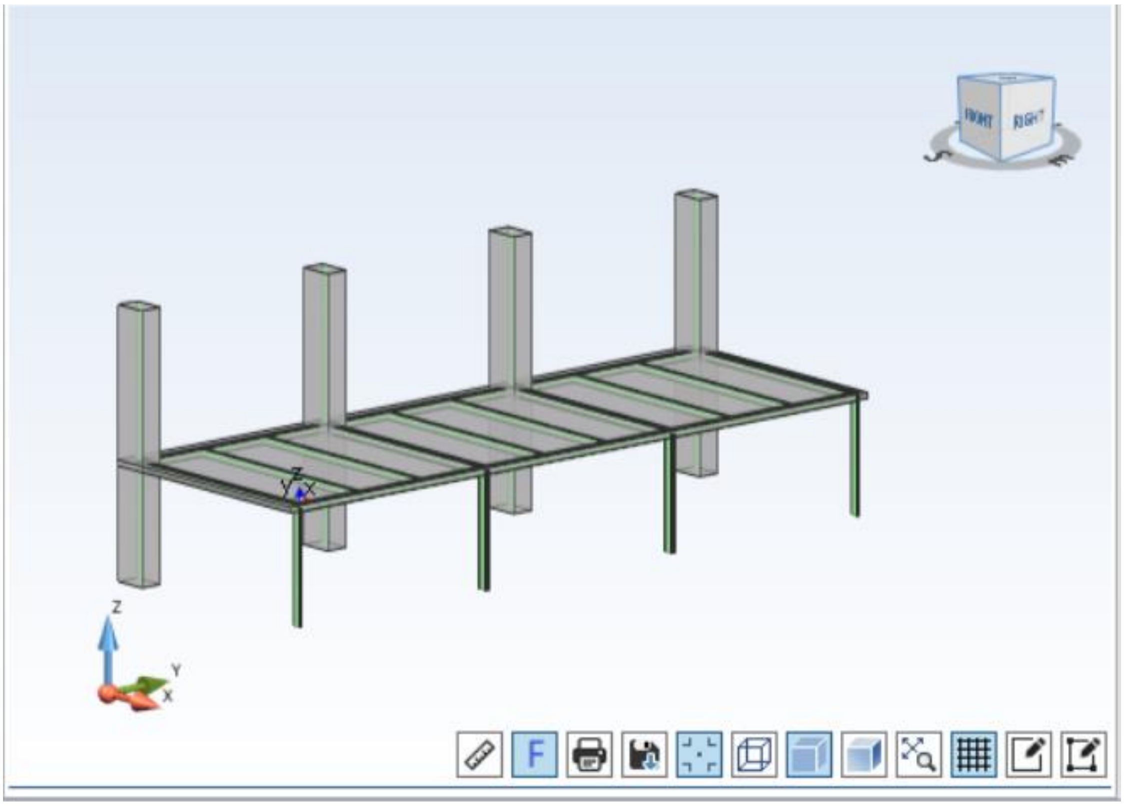
Five types of concrete elements can be calculated using the Advance Design modules: footings (continuous and isolated), beams, columns, slabs, and walls.




FRONT
Z
Y
X


FRONT
Z
Y
X

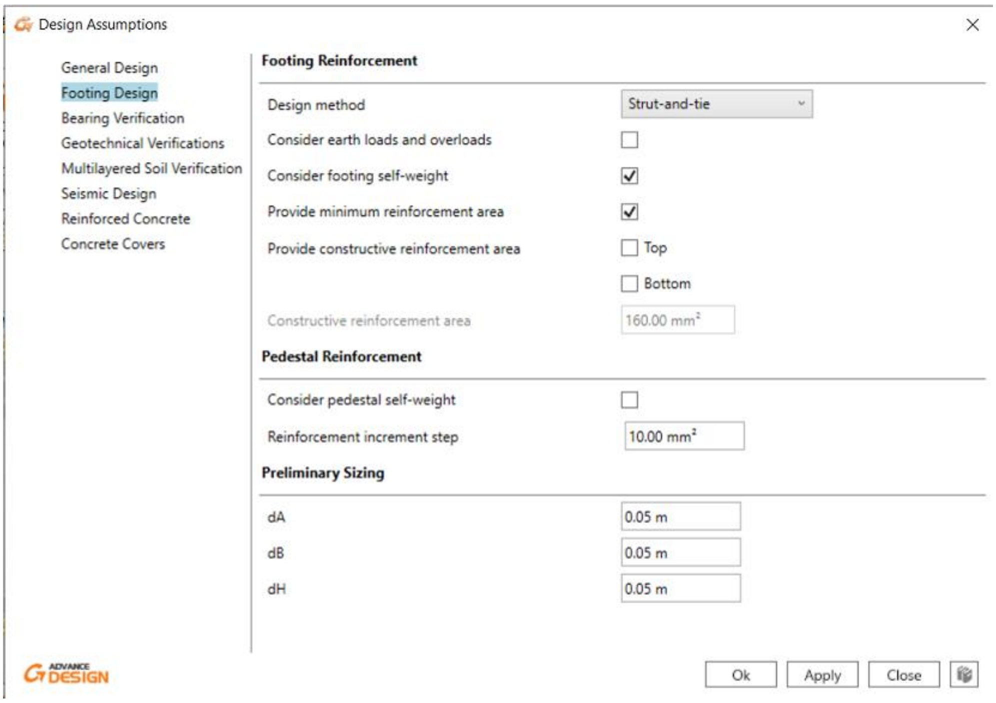




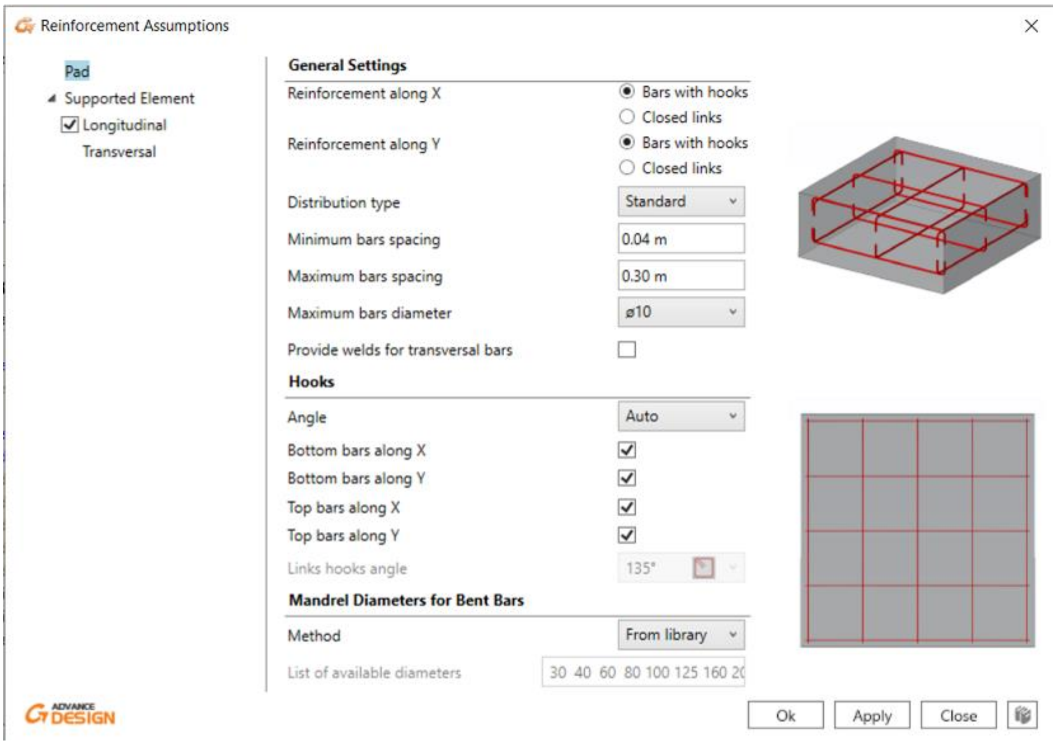
The unavoidable step before running a structural design session is to define the structural design assumptions.

Structural design assumptions can be split in two different families:

✓ Design assumptions (Design Settings)

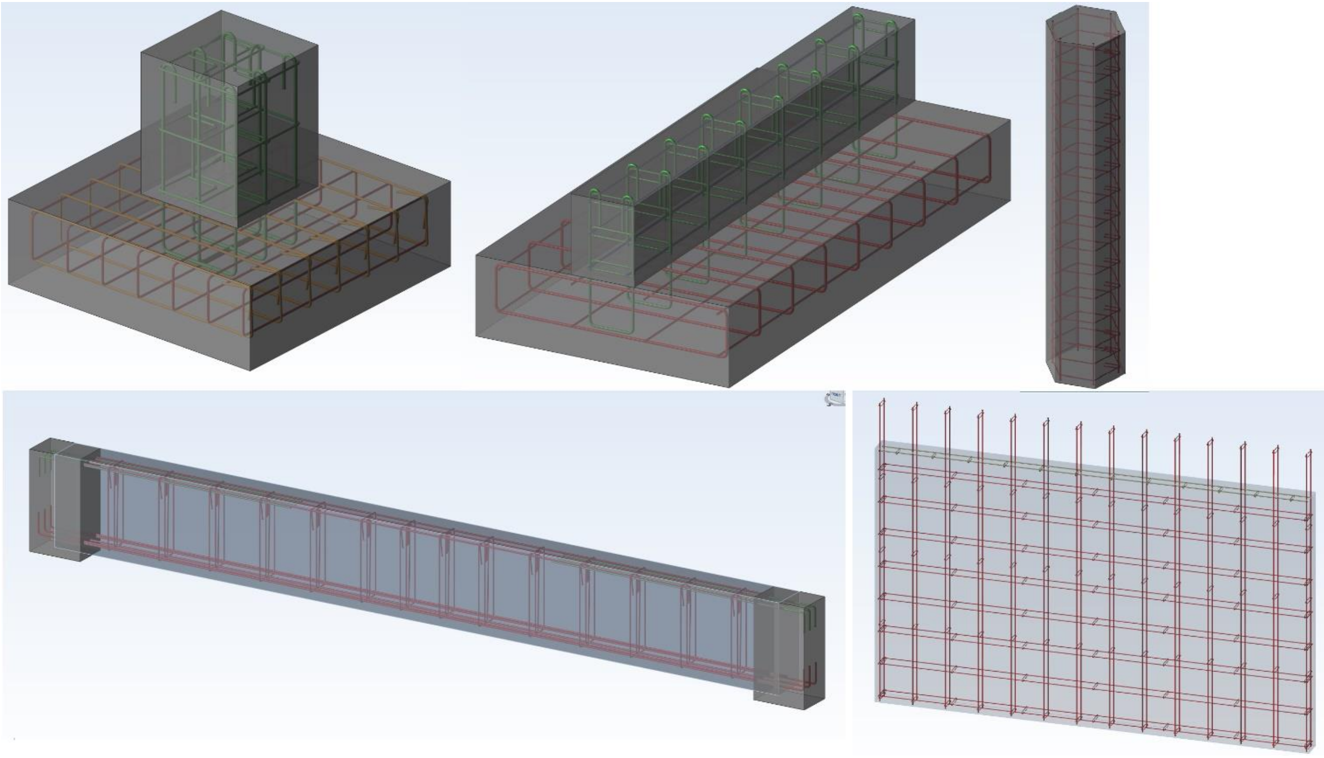


✓ Constructive dispositions for the automatic placement of bars inside concrete members (Reinforcement Settings)



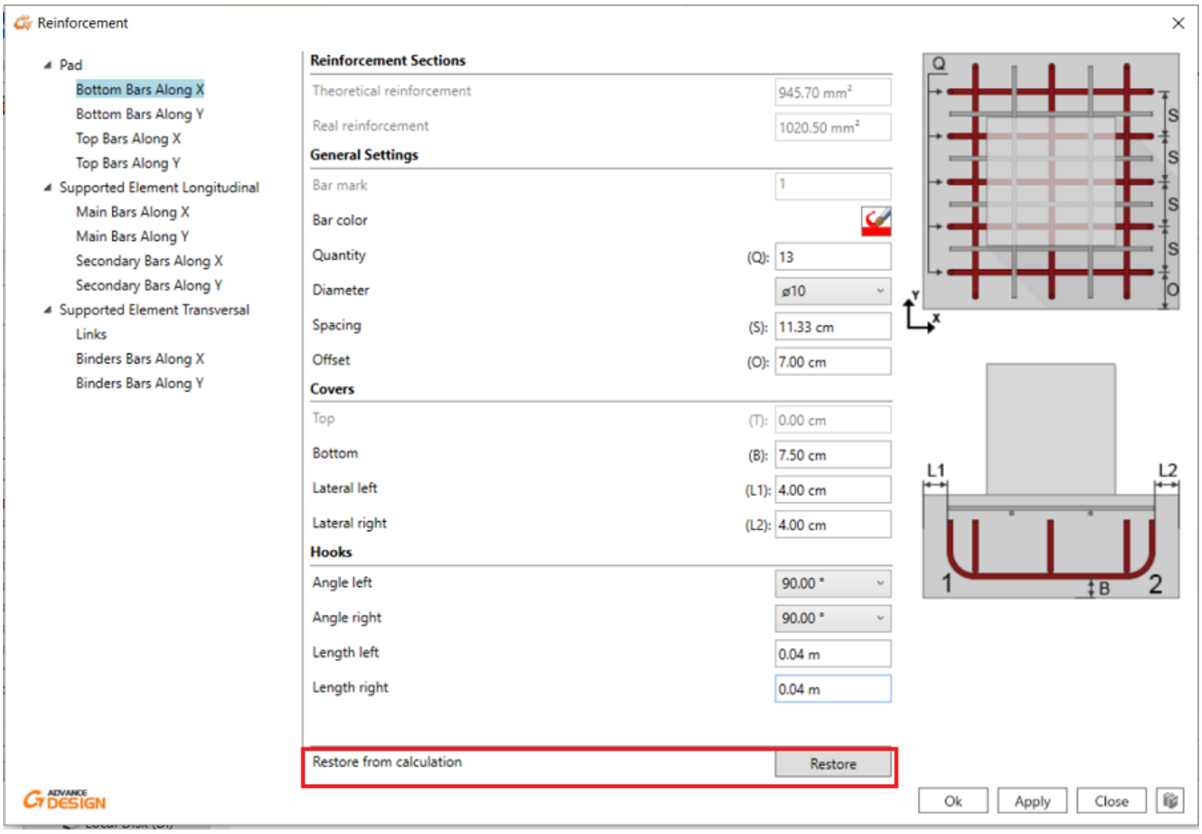
Both assumption families (Design and Reinforcement) are country-code dependent (Eurocodes with different national appendixes, US codes, Canadian codes, NTC codes).

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.

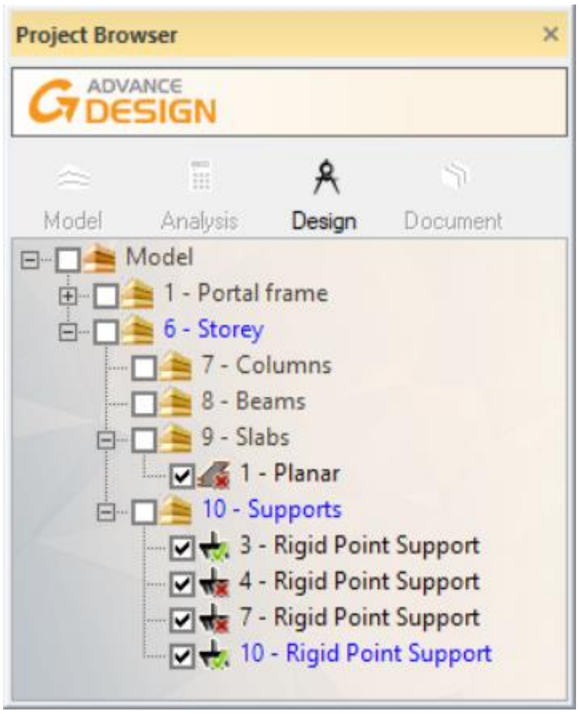


After the reinforcement is generated, you still have the option to adjust it. The reinforcement can easily be modified, or new bars can be added, with the possibility to see if the area of modified bars exceeds the theoretical area calculated by the Advance Design module.

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



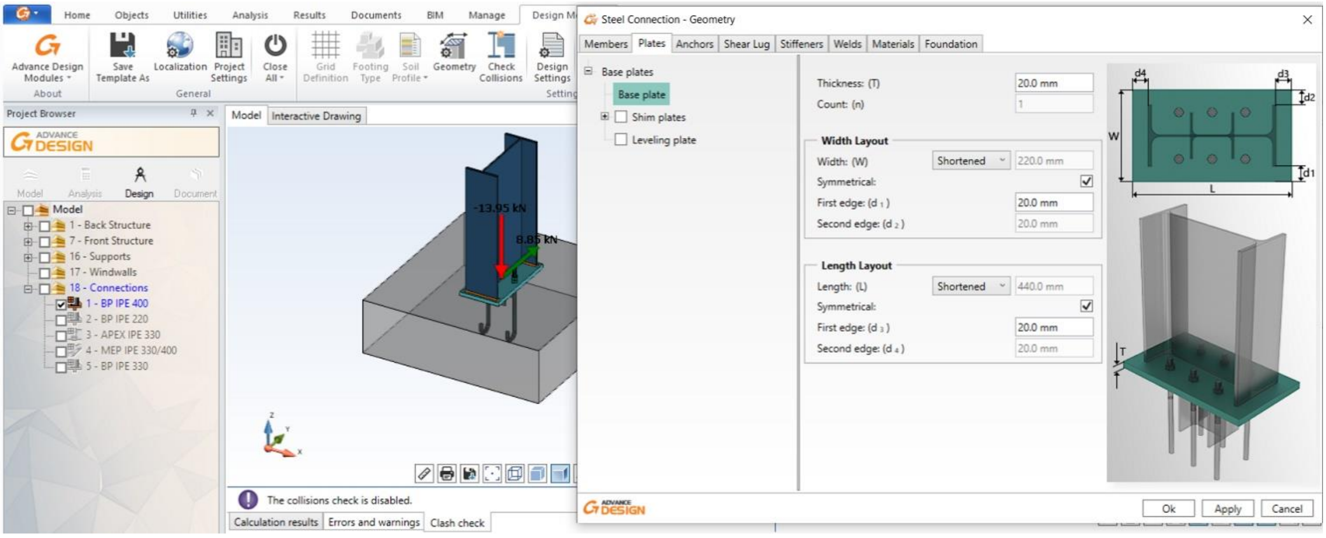
The status of calculation results is displayed in the Design mode for each separate element after performing the calculation. A green check sign will appear for the calculated element that has no error message or failed verifications, which means that the obtained maximum work ratio for that element 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.



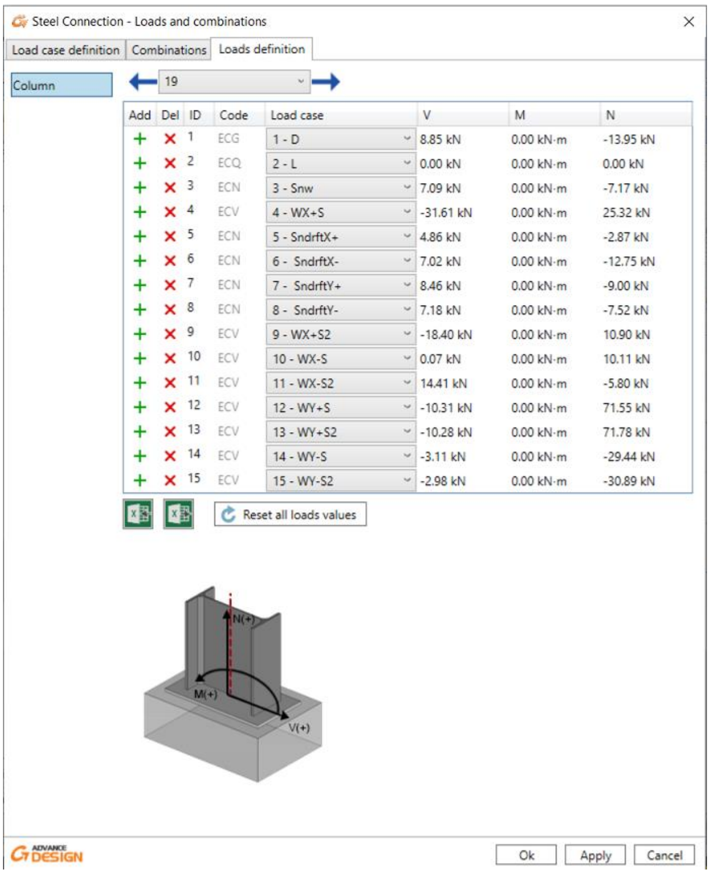
Steel Connections

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 main members of 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.

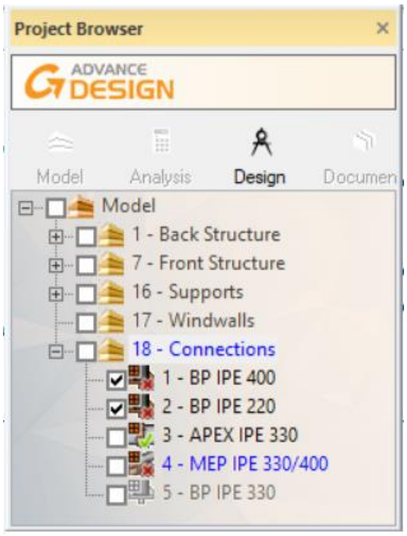


The efforts from Advance Design are transferred as envelope of efforts for a simple reason: for steel structure there are too many load cases\combinations and the calculations take too long. Using the envelope, the calculation is done in just few seconds.



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.



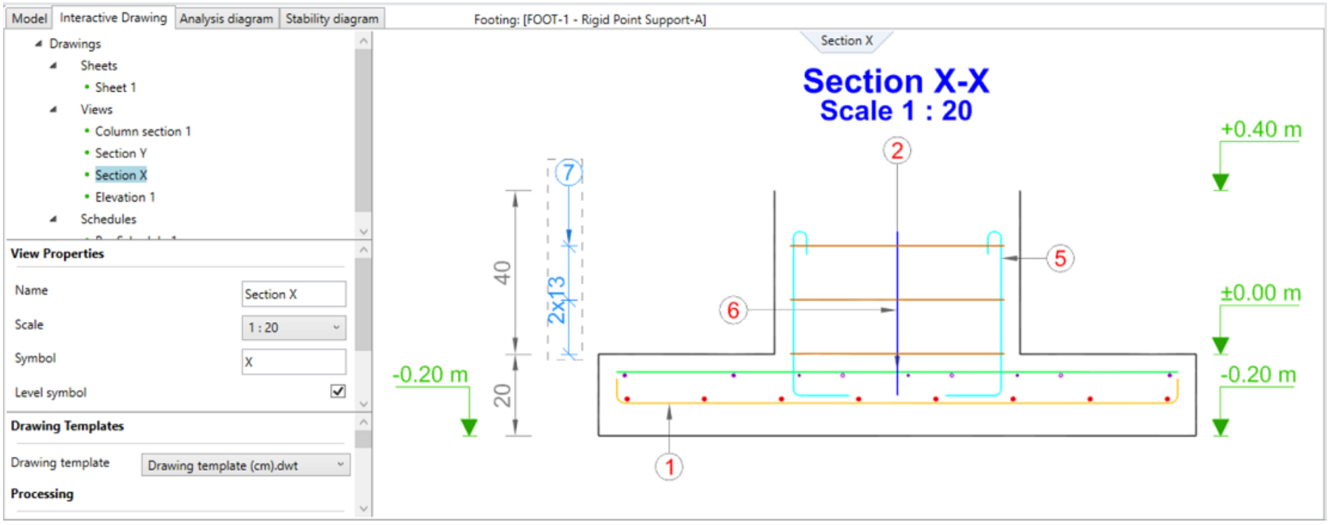
Output: Calculation Results Tab

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	116: 1.35x[1 G]+1.5x[2 Q]+0.75x [3 N]	393.83 kN	363.08 kN	108.47%
Sliding	113: 1x[1 G]+1.5x[2 Q]	38.32 kN	130.02 kN	29.47%
Overturning	101: 0.9x[1 G]	3.08	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 mm²	550.71 mm²		87.69%
Top Along X	352.10 mm²	312.10 mm²		88.64%
Top Along Y	301.80 mm²	78.03 mm²		25.85%

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

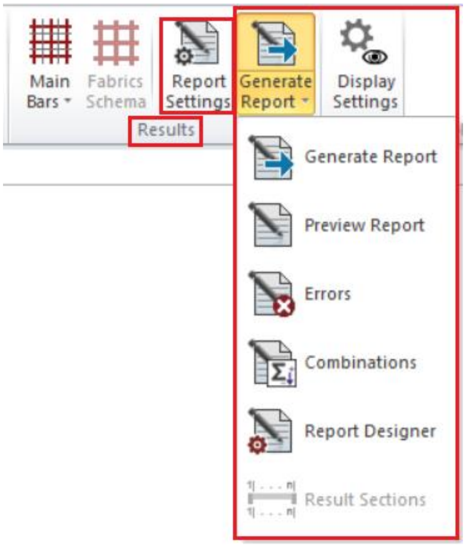
On the drawing, the program automatically creates a bar schedule and a title block with all the information about the project and the element (this information will be imposed in ‘Drawing Settings’ from Advance Design modules).



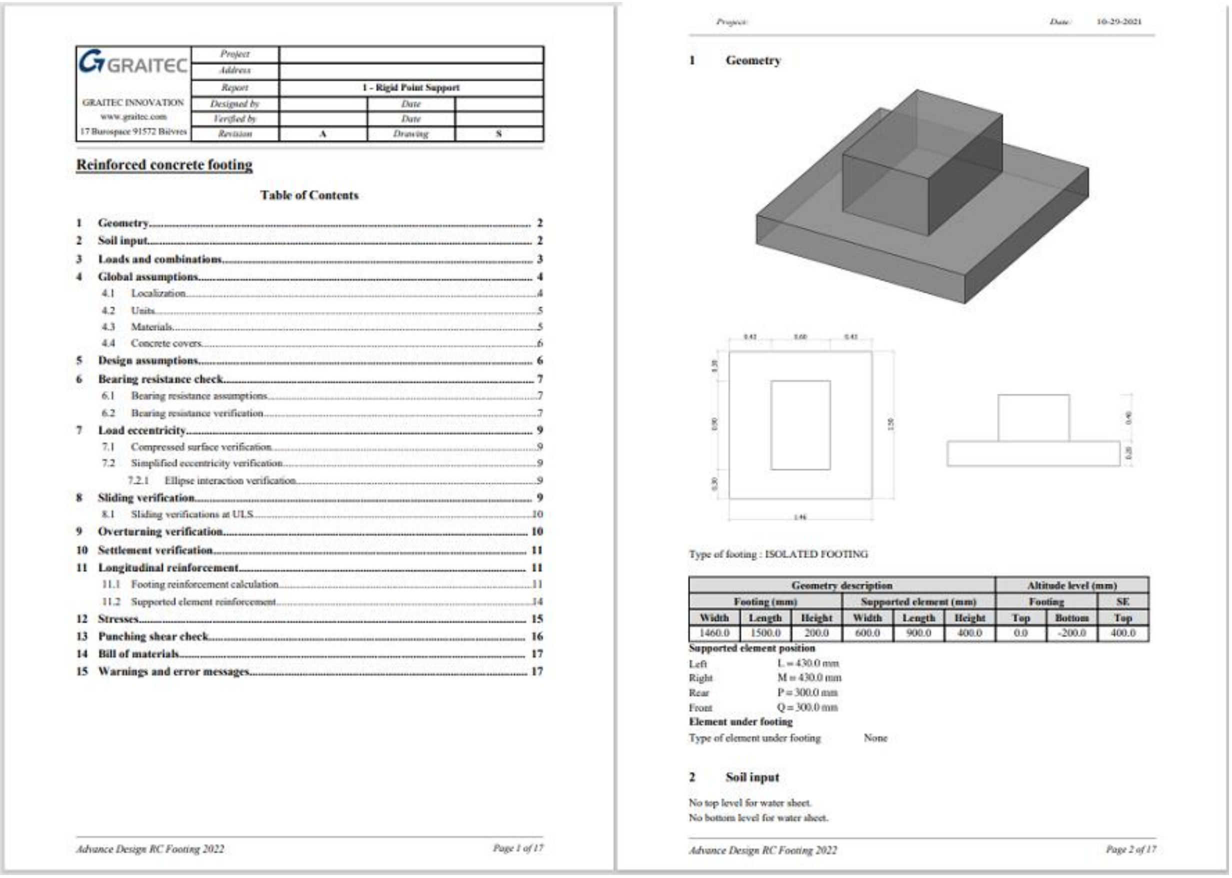
You can manage all drawing components in the same window. For example, you can quickly add any number of new views to a given drawing

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.



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



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