

GRAITEC

advance

DESIGN

STARTING GUIDE



TABLE OF CONTENTS

INTRODUCTION.....	3
Welcome to Advance Design.....	3
About this guide.....	4
Where to find information?	4
Contact the technical support	4
WHAT IS ADVANCE DESIGN?	5
INSTALL ADVANCE DESIGN	6
System requirements	6
Hardware configuration	6
Software configuration.....	6
Advance Design installation	6
STARTING ADVANCE DESIGN	7
ADVANCE DESIGN ENVIRONMENT	8
MODELING: CREATE THE DESCRIPTIVE MODEL.....	11
Element types.....	11
Input of elements	12
Example: Input structure elements.....	12
Definition of elements properties	13
Example: Define the elements properties	13
Systems of elements	14
Example: Create systems of elements.....	14
CAD functions	15
Example: Copy elements	15
Generate Loadings	16
Example: Generate loadings.....	17
Example: Create loads combinations	18
Define analyses.....	18
Example: Define a modal analysis.....	19
Model verification	19

ANALYSIS: MESHING AND CALCULATION	20
Create the analysis model	20
Meshing	21
<i>Example: Define the model meshing</i>	21
Calculation	22
Finite elements calculation	22
Reinforced concrete calculation.....	23
Steel calculation.....	23
<i>Example: Run a complete calculation sequence</i>	24
RESULTS EXPLOITATION	25
Graphical visualization of results	25
<i>Example: Create a graphical exploitation of FE results</i>	26
Result curves	28
<i>Example: Display result curves on a section cut</i>	28
Stresses diagrams	29
<i>Example: Display a stresses diagram</i>	29
Post-processing animation	30
<i>Example: Create a postprocessing animation</i>	30
Expert Design postprocessing	31
Reinforced concrete results	31
<i>Example: View longitudinal reinforcement on linear elements</i>	31
<i>Example: View reinforcement results on a column</i>	32
Steel results	33
<i>Example: Verify the steel elements stability</i>	33
Steel elements optimization.....	33
<i>Example: Optimization of steel shapes</i>	34
Memorized exploitations	34
<i>Example: Create an exploitation view</i>	35
Reports	35
<i>Example: Generate a report</i>	36

INTRODUCTION

Welcome to Advance Design

From the elements modeling to the structure calculation, results exploitation and structure optimization, Advance Design offers a complete environment for the static and dynamic analysis of 2D and 3D structures with the finite elements method.

This software also provides advanced design capabilities for steel and reinforced concrete structures. The verification of steel elements starts from an initial dimensioning and may continue by 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 analysis modules
- New software technology, for example:
 - Possibility to record video sequences of structural distortion
 - A calculation engine that adjusts itself to complex geometries
 - The Result Memory technology that allows the automatic real time update of results exploitation with each calculation sequence
 - Generate calculation reports with real script recording including graphics that can be replayed during iterations

About this guide

This guide's purpose is to describe the main functionalities and interface of Advance Design, and also to make you familiar, by a few small examples, with the program's working process. The examples follow each description of Advance Design functions.

This guide contains a general presentation of Advance Design; therefore not all of its features are described here. For detailed information concerning all commands and functions of the program, please refer to the *Online Help* which is accessible from Advance Design interface.

Where to find information?


Advance Design has an online help system that offers you detailed information and step-by-step instructions for every function.

The online help is available from the Advance Design interface with the help of the following commands:

- From menu: choose ? > **Online help ...**
- From keyboard: press **F1**

Contact the technical support

Graitec technical support is available by phone, fax or email. You can write to Graitec team using the following commands:

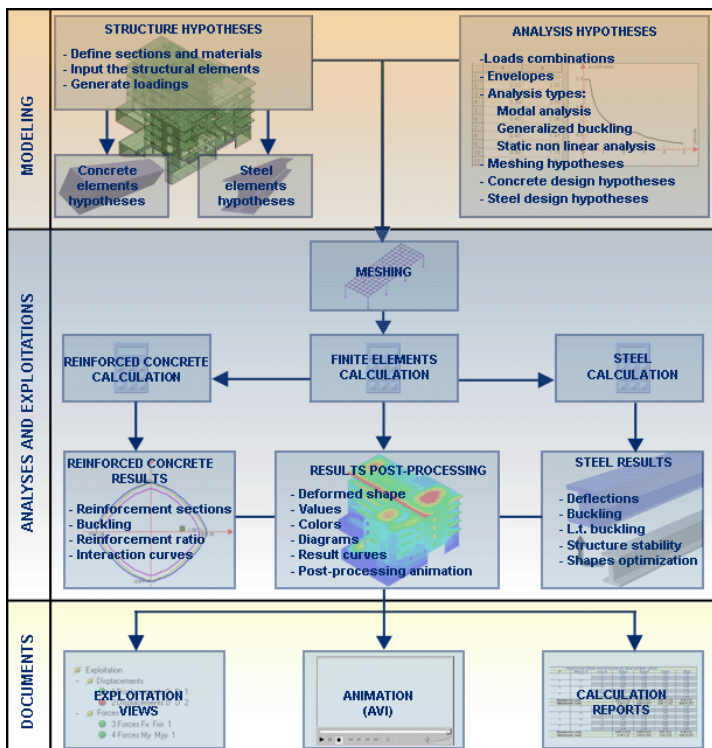
- From **Standard** toolbar: click on  to send an email to Graitec
- From menu: access ? > **Technical support**. A template email is sent to the technical support team who will quickly solve your problem and give you a precise answer. The model of the current project is automatically archived and attached 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 results exploitation and structure optimization.

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

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



INSTALL ADVANCE DESIGN

System requirements

Hardware configuration

- A PC computer or compatible equipped with a Pentium IV processor (or equivalent)
- 512 MB RAM memory (1024 MB recommended)
- At least 150 MB space available on the hard drive for installation
- A CD-ROM reader
- Windows compatible graphics adapter (128 MB video RAM recommended)
- Windows compatible printer or plotter with supplied drivers
- Mouse

Software configuration

- Operating System: Windows 2000 Pro or XP Pro (or newer)
- Screen resolution: 1024 x 768
- 24 bit color palette recommended

Advance Design installation

Before installing Advance Design, you must:

- Under Windows 2000 or XP, make sure that you have administrator rights
- Close all opened applications
- Make sure that the provided security key is plugged in to the computer

Please proceed with the installation as follows:

1. Insert the Advance Design CD-ROM in the reader
2. The setup program starts automatically. From the starting screen, click on "SETUP" to launch the installation

Note: *If the starting screen doesn't appear automatically after introducing the CD-ROM in the reader, double click on "Setup.exe" to launch the installation.*

Follow the steps proposed by the installation wizard. Click on "Next" to continue.

3. Read the license agreement. Click on "Yes" to accept the terms and continue the installation
4. In the following screen, if you want to change the installation path, click on "Browse". Click on "Next" to continue.
5. Wait a few moments while the Advance Design is installed on your computer. Click on "Finish" when the installation is complete.

The utilization of Advance Design is possible only if the security key provided with the software package is correctly installed and configured. After the installation, it is necessary to configure the program's security settings:

1. From Windows **Start** menu, access **Programs > Graitec > Advance Design > Security**
2. The "Security" window is displayed. Select from the configuration field the type of protection that you acquired
3. Click on "OK" button to apply and exit the window

For details regarding all security key types and the corresponding installations, please refer to the Advance Design User's Guide.

STARTING ADVANCE DESIGN

You can launch Advance Design using various methods:

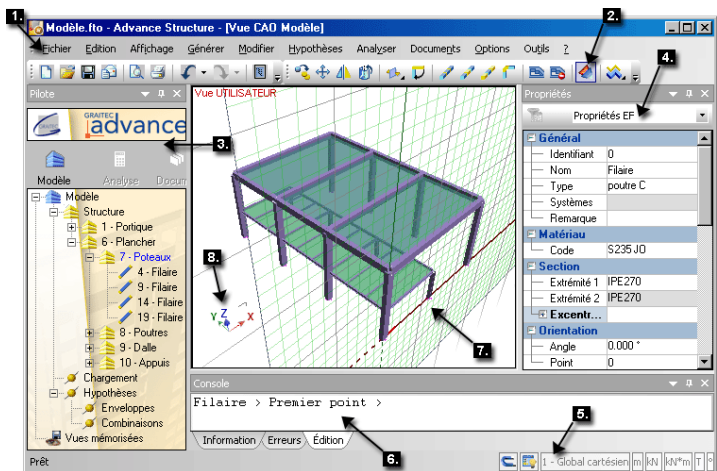
- From Windows **Start** menu, choose **Programs**. Select **Graitec** menu and click on the Advance Design sub-menu.
- You can also double-click on the Advance Design icon on your desktop.

To start another work session simultaneously:

- Double-click on an existing .fto file from its disk location
- Double-click on the Advance Design icon on your desktop

ADVANCE DESIGN ENVIRONMENT

Advance Design offers a complete environment for modeling, analysis and results exploitation - all fully integrated in the same interface.



1. Menus


You can access the program commands scrolling the drop-down menus on the menu bar. The menus are ordered from left to right considering the project steps.

2. Toolbars

The different types of commands are grouped in toolbars, which you can easily display and position (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 Hypotheses**; **Analysis - F.E. Results** etc) are automatically displayed or hidden, in order to optimize the workspace during the work process.

3. Pilot

The pilot represents the control center of Advance Design that regroups all the components of the model as an arborescence of elements and allows an easy access to the project's working modes: "Model", "Analysis" and "Document".

You can easily change the working mode by a click on the corresponding icon placed on the upper side of the pilot: .

Each item of the pilot has a context menu that gives access to different commands specific to the current step of the project. The pilot contents differ for each working mode:

- **Model mode:** regroupes the structure elements, which you can organize in systems and sub-systems; loadings (organized in family of cases and load cases), analyses hypotheses (loads combinations, envelopes, analyses types), saved model views.
- **Analysis mode:** allows the management of analysis cases; analysis types; saved analysis and exploitation views.
- **Document mode:** gives access to all documents generated during the work process: reports, memorized views, AVI files.

4. Properties window

You can view and modify the attributes of all the model's entities in the properties window. The properties are displayed in a tree representation by various categories. The properties window is displayed dynamically, when an element (or a drawing tool) is selected, and allows the access to the common properties of a selection of elements of the same type.

5. Status bar

The status bar displays information regarding the program's status during different phases of the project. It also contains buttons that give access to the configuration of certain parameters: snap modes, objects tooltips content, current coordinate system, working units.

6. Command line

The command line is the dialog area between the user and the program. It contains three tabs, specialized in different operations:

- *Information:* displays status of the current operations
- *Edit:* intermediates the dialog between user and application, allowing you to draw / modify objects typing parameters in the dialog area of the command line
- *Errors:* displays warnings and error messages

7. Graphic area

Representing the design area of the application, it allows an easy and intuitive use of CAD commands and a realistic rendering of the model. Each element of the graphic area has a context menu that allows quick access to different specific commands (select, generate elements on selection, display / hide elements, etc).

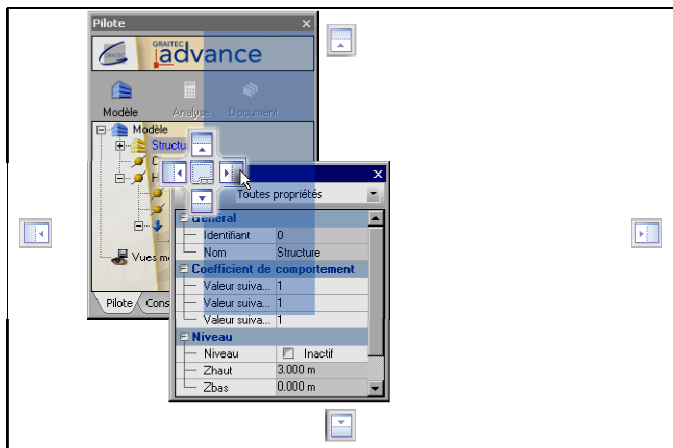
You can also split the graphic area in several viewports (from one to four); each of these can have different display settings and point of view (zoom on a certain part of the structure, realistic or simplified rendering, etc.).

The default workplane of the drawing area assists you in the structure modeling. It is possible to define the workplane's parameters and you can hide or display it easily during your work process.

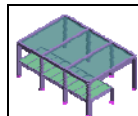
8. Coordinate system

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's interface is intuitive, enhanced for an easier manipulation of its different components and commands (advanced docking, undocking, auto-hiding, tabbing etc).



MODELING: CREATE THE DESCRIPTIVE MODEL

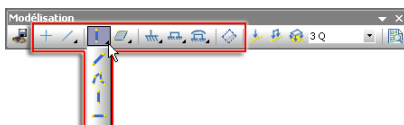


The structure modeling can be done entirely with the use of various CAD tools via the graphic area, where a 2D or 3D representation of the model can be visualized at any time.

The various zoom and view commands (rotate around the model, predefined views...) allow a fast and easy manipulation of the graphical elements.

Element types


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



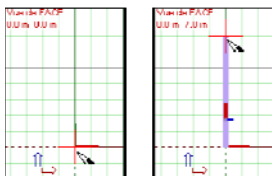
Example	Element type
	Linear elements (bar, beam, short beam, variable beam, tie, gap, cable)
	Planar elements (membrane, plate, shell, plane strain)
	Supports (point, linear and planar supports, which can be rigid, elastic or traction only / compression only)
	Windwalls: elements that serve for the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

Input of elements

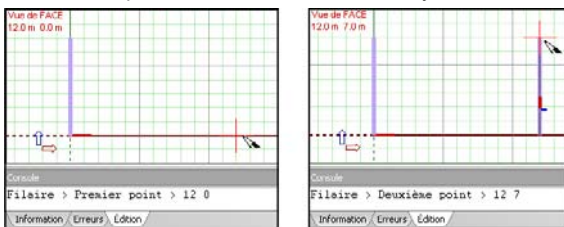
You can draw elements in the graphic area using the keyboard (typing coordinates in the command line) or the mouse, relatively to workplane's points or to existing entities. Advance Design also provides various automatic drawing tools (generate elements on selection, portal frames and vaults generators etc.)

 **Example:** *Input structure elements*

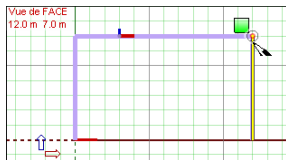
1. From **Modeling** toolbar: click on 
2. In the graphic area (XZ plane): click to define the column on 0 0 coordinates for the first extremity and 0 7 for the second one.



3. With the linear element drawing tool still active, type in 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, with the help of "Extremity" snap mode:




- Draw a column of 3 meters height placed on 6 0 XZ coordinates:



- Create the storey beam: with the linear element drawing tool still active, press **Alt + S** to access the snap modes dialog box; select the perpendicular snap, then draw the beam as shown below:




- From **Modeling** toolbar: click on  icon and, in the drawing area, click on the bottom extremity of each column to create supports:

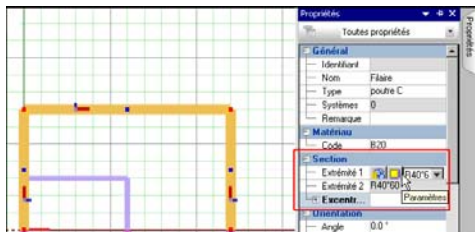



Definition of elements properties

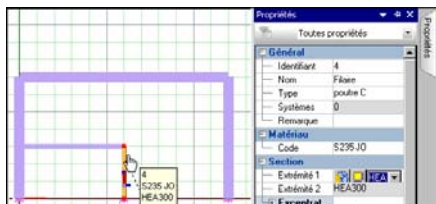
You can define the attributes of each type of element in the properties window (name, ID, different parameters). By default, the properties window appears when an element is selected and auto-hides when it's empty.

 **Example:** Define the elements properties

- Click on the two columns and beam of the main portal frame to select them.
- The properties window containing their common attributes appears; in "Material" field select **B20** and "Section" field define a **R40*60** section type:



3. Select the storey column, and in its properties window: choose a **S235JO** material. To define the element's section: click on  button to access the sections library and select from European Profiles - **HEA300**:




4. Proceed in the same way to define the material and section for the storey beam: **S235JO - HEA220**.

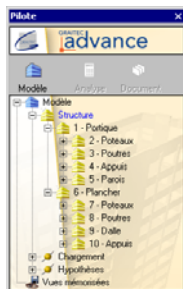
Systems of elements

The structure elements that you create (and also geometrical elements, help entities etc.) are stored in the pilot, in **Model** mode. The pilot's context menus for each of its items allow a fast access to different modeling commands and an advanced management of the elements (hide / display, select, delete, group in systems etc.).

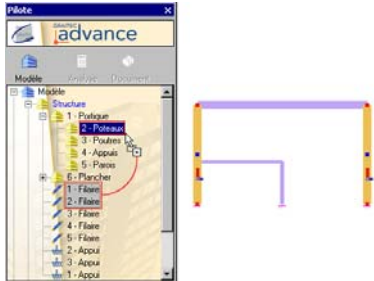
The system concept helps you define the behavior of different groups of elements, for example, assign design templates; you can also easily perform different operations on a group of elements using their system's context menu commands. The level function available in the system's properties window allows the definition of its level settings and thus create faster and easier structure elements on different altitudes (storeys): a column by one click, a wall by two clicks.

 **Example:** Create systems of elements

1. In the pilot: select "Structure", right click and choose from context menu **Systems management > Create a subsystem**
2. Type the system's name: "Portal frame"
3. Select "Portal frame" system, and using the steps described above, create the following subsystems: "Columns", "Beams", "Supports", "Windwalls"
4. In the same way, create under "Structure" the "Storey" system with the subsystems Columns", "Beams", "Slab" and "Supports"



5. Select the two columns of the portal frame and, in the pilot, drag-and-drop the selected elements to the "Portal frame" - "Columns" system
6. Using the same method, place all the elements of the model in the corresponding systems



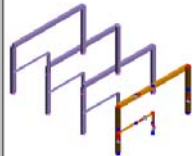
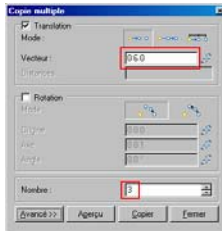
CAD functions

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



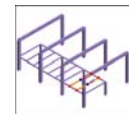
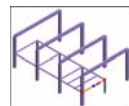
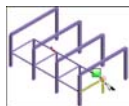
Example: Copy elements

1. Define a 3D view of the workplane: click on icon from **Predefined views** toolbar (or press **Alt + 6**)
2. Press **Ctrl + A** to select all the elements of your model
3. Right-click in the drawing area and select "Copy" (or press **Insert** key). The "Multiple copy dialog box" appears, allowing you to define the copy parameters:
 - Copy by "Translation"
 - Copy vector: 0 6 0
 - No. of copies: 3
4. Click on "Preview" button to preview to copy
5. Click on "Copy" button to execute



6. To create the rest of the storey elements:

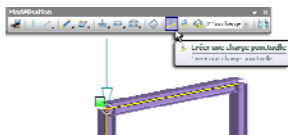
- ✓ In the pilot: select the "Storey" - "Beams" subsystem
- ✓ Select the linear elements drawing tool and draw two longitudinal beams (**S235JO** with **HEA220** section)
- ✓ Select the first transversal beam and make 2 copies in 0 2 0 direction.
- ✓ Select the two copied beams and define, in their properties window, **S235JO** material and **IPE200** section
- ✓ Make 2 copies of the selected beams in 0 6 0 direction



Generate Loadings

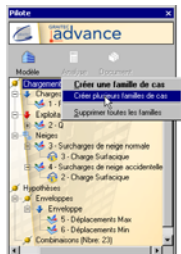
You can generate and organize loadings with the help of the pilot. Loads are grouped in the pilot under "Loadings" in load cases (self weight, static, seismic...) and case families (permanent loads, exploitation loads, snow, wind, temperatures...). Each case family may contain several load cases, as each load case may contain several loads.

You can generate loadings via the graphical input, using the load creation tools available from **Modeling** toolbar, pilot or menus. You can also use the automatic tools (pressure loads generator, climatic loads generator, loads on selection, etc.).


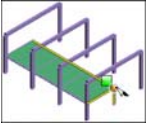

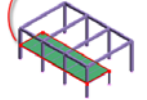
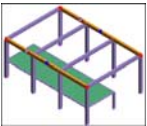


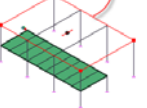


You can define the parameters of loads, load cases and case families in their properties window. You can also manage the loadings using their context menu commands from the pilot.

After you have defined the loadings, you can create load combinations and envelopes (using the pilot or the **Analyze** menu commands). The combinations manager allows you either to manually create combinations, or to load regulatory combinations available in Advance Design.

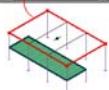


 **Example:** *Generate loadings*

1. Generate the self weight: select from menu **Generate > Load > Self weight**. The "Permanent loads" family and a self weight load case are automatically created in the pilot.
2. Create an exploitation load:
 - Create first the storey slab. In the pilot: select the "Storey" - "Slab" subsystem. Click on  icon from **Modeling** toolbar and draw the slab as shown. In the slab's properties window, define a 15 cm thickness. 
 - In the pilot: right click on "Loading" and choose from context menu "Create a case family". In the displayed window, select "Exploitations" and press "OK". An exploitation family and a corresponding case are created in the pilot. 
 - Select the exploitation case **2Q** in the pilot and, in the drawing area, select the storey slab; right click and choose from context menu "Loads / selection". The planar loads properties window is automatically displayed: input here the loads intensity on FZ: **-5 kN** and press "OK". 
3. Generate snow loads. Create at first two beams and a windwall on the portal frame:
 - In the "Portal frame" - "Beams" subsystem create two longitudinal portal frame beams, using **B20** material and **R40*60** section type. 
 - In the pilot: select "Portal frame" - "Windwalls" subsystem. Select the two longitudinal beams; right click and choose from the context menu "Windwalls on selection". From **Rendering** toolbar, click on  icon to enable the "Axes" rendering mode; this allows you to view the direction of span on windwalls. In the windwall's properties window: set the span direction towards the longitudinal beams, considering the windwall's local axes. 


5. Generate the snow loads:

- In the pilot: right click on "Loading" and generate a snow family using the same steps described above. A snow family and a snow case are created in the pilot.
- From the snow family properties window, select **1B** snow region.
- To automatically generate the snow loads on the windwall, select from menu **Generate > Load > Climatic loads**.



Example: Create loads combinations

1. In the pilot: right-click on "Combinations" and choose from the context menu "Properties". The "Combinations" dialog box appears
2. Click on "Load" button, select in the displayed window the **BAEL91** combinations file and click on "Open". The combinations defined after the BAEI codes are automatically generated.
3. Repeat the operation to generate combinations with **CM66** codes. You can view the number of created combinations in the pilot.



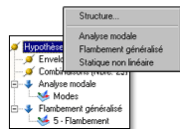
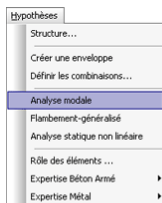
Define analyses

During the modeling step, Advance Design allows you to define several types of analyses (modal, buckling, static nonlinear), and also the concrete and steel design hypotheses.

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

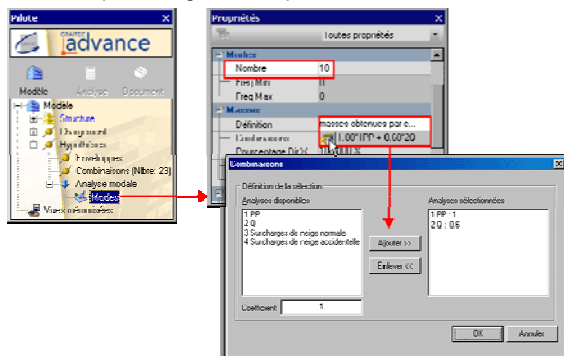
You can also use the pilot's commands to manage analyses. The pilot allows you to view and select the analyses that you have created.

You can define the analyses cases parameters in the properties window.



Example: Define a modal analysis

1. From menu: select **Hypotheses > Modal Analysis**
2. In the pilot: notice that a modal analysis family and the "Modes" case are placed under "Hypotheses" group. Select the "Modes" case to display its properties window
3. Define the modes parameters:
 - ✓ Input the number of vibration modes: **10**
 - ✓ Masses definition: select from the combo-box "masses obtenues par c..." field below, click on icon to access the masses combinations dialog box. Define here the following combination: **1*1PP + 0.6*2Q**
 - ✓ For Mass percentage on Z: input **0**.

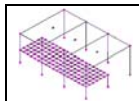


Model verification

At any time during the modeling step, you can verify the model's coherence and integrity with the help of verification function. Access the command

Analyze > Verify, or simply click on icon from the **Modeling** toolbar. If there are errors or warnings, these are displayed in the command line.


ANALYSIS: MESHING AND CALCULATION

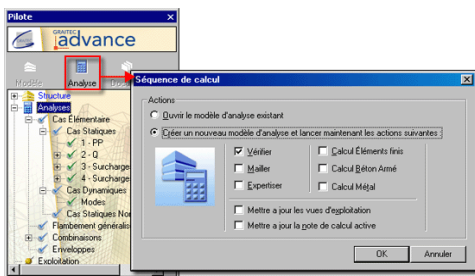


In the next step, after the verification of the model's coherence and validity, the program creates the analysis model. In this phase, you can run the structure meshing and the model calculation considering the defined analyses (finite elements calculation and concrete / steel verification).

Advance Design allows you to easily define the desired sequence of operations to perform in one analysis iteration (verification, meshing, calculations...).

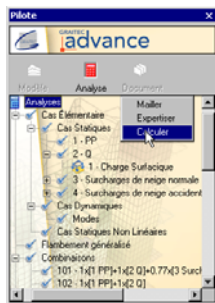
Create the analysis model

In order to mesh and calculate the structure, it is necessary to create the analysis model. Once you have verified the model's validity, simply access the command **Analyze > Create the analysis model**, or, in the pilot, click on  icon.



An assistant helps you concatenate the desired operations (verify, mesh, finite elements calculation, reinforced concrete calculation, etc) into a calculation sequence which is performed automatically.

You can control and view the analysis model's components in the pilot, in **Analysis** mode. The context menu commands available for each element of the pilot help you easily manage the analysis operations.



After the analysis model creation, new toolbars and commands are available (**Analysis - Hypotheses** toolbar, for example), while the modeling tools are inactive.



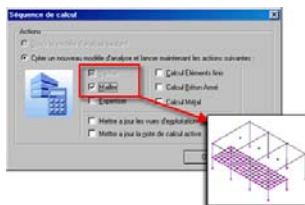
Meshing

Advance Design allows you to choose between two different mesh engines: "Grid" and "Delaunay".

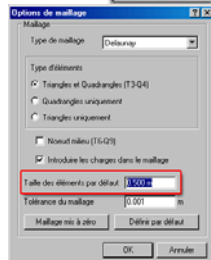
Therefore, the finite element meshing is done taking into account the global mesh settings (defined via the command **Options > Mesh...**) and also the mesh parameters defined for each element (available in the properties window). It is possible to define the mesh parameters of each element using the simplified method (a meshing density along each of the local axes), or the detailed method (a meshing density for each of the element's sides).

Example: Define the model meshing

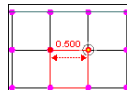
1. In the pilot: click **Analysis** icon to access the "Calculation sequence window"
2. Select "Mesh" and click "OK"; Advance Design creates the analysis model and automatically performs the model meshing.



3. Modify the mesh density:
 - ✓ Choose from menu **Options > Mesh...** to display the "Mesh options" dialog box
 - ✓ In the "Default element size" field: input 0.5 meters
 - ✓ Click "OK" to apply and close

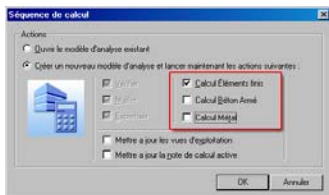


4. Recreate the mesh: from **Analysis - Hypotheses** toolbar, click on icon. The meshing is modified according to the global settings.



Calculation

After meshing, Advance Design is ready to calculate the model. The "Calculate" command gives access to "Calculation sequence" window, that helps you select the calculations to perform:

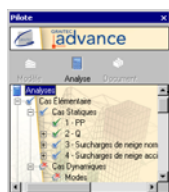
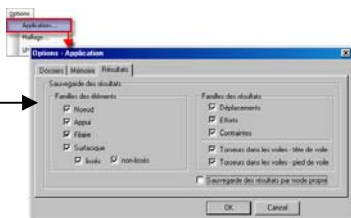


Finite elements calculation

A powerful finite elements engine performs the model calculation, considering the structure hypotheses:

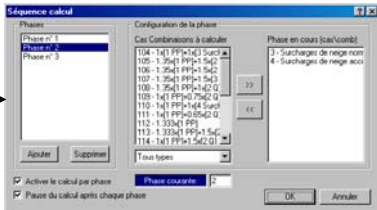
- Defined analyses (static and dynamic calculation, linear and non-linear analyses, large displacements, generalized buckling...)
- Finite elements parameters of structure elements (defined in the properties window)

Before calculation, you have the possibility to define the elements to calculate and the type of results that you want to obtain, in order to improve the calculation speed and the memory usage.



During the analysis step you have the possibility to decide which analyses to calculate (using the pilot commands).

Advance Design allows you to group analyses in calculation phases and calculate them step by step (allowing modifications of properties for each phase).



Reinforced concrete calculation

The reinforced concrete engine calculates the reinforcement of concrete elements by serviceability limit states (ELS) and ultimate limit states (ELU and ELUA) and verifies the concrete sections by interaction curves.

The reinforced concrete calculation is possible only if you have created the regulatory load combinations and you have run the finite elements calculation. The reinforced concrete calculation takes into account the global and local concrete design hypotheses:

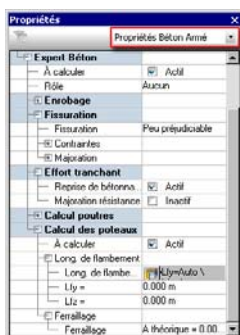
Global concrete hypotheses

Concern the calculation methods of reinforced concrete, the columns verification, reinforcement and buckling parameters etc.



Local concrete hypotheses

You can define the local concrete design hypotheses in the properties window of the concerned elements.



Steel calculation

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

As well, the steel calculation is possible only if you have created the regulatory load combinations and you have run the finite elements calculation.

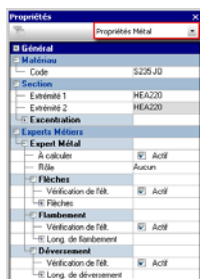
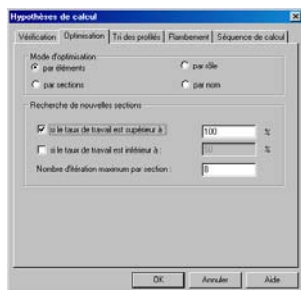
The steel calculation takes into account the global and local steel design hypotheses:

Global steel hypotheses

Concern the steel calculation methods, the optimization criteria, the buckling calculation methods, etc.

Local steel hypotheses

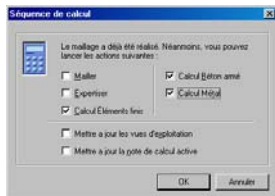
You can define the local steel design hypotheses in the properties window of the concerned elements.



Note: After the calculation, you can view the results and modify the elements parameters, if necessary. You can iterate thus the desired calculations until you obtain the appropriate results.

 **Example:** Run a complete calculation sequence

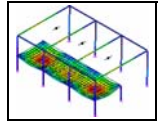
1. Select from menu **Analyze > Calculate**
2. In the "Calculation sequence" dialog box, check "Finite elements calculation", "Reinforced concrete calculation" and "Steel calculation"
3. Press "OK" to launch the selected operations.



The command line displays the performed operations, and informs you when the calculations are done.



RESULTS EXPLOITATION

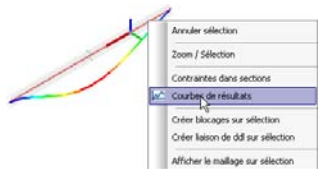


The phase following the model calculation, also called the post-processing step, allows you to view the results on the graphical model or as calculation reports, result curves on the selected elements etc.

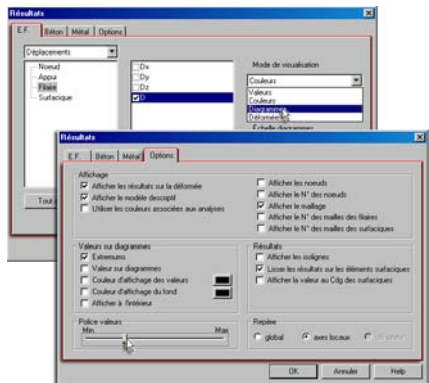
Graphical visualization of results

A new set of tools and commands are active during the results exploitation step. They allow you to select the most convenient mode to display the desired results. You can access the results visualization commands:

- From results toolbars, which appear automatically once the corresponding calculation is done.
- From element's context menu: Advance Design allows you to view results on selection in the graphic area. When you haven't defined a selection, the results are displayed on the entire structure.




- Using the results configuration dialog box, that allows a detailed configuration of results display. You can choose between different visualization modes: colors, values, deformed shape, iso-values, iso-regions, vectors....




 **Example: Create a graphical exploitation of FE results**

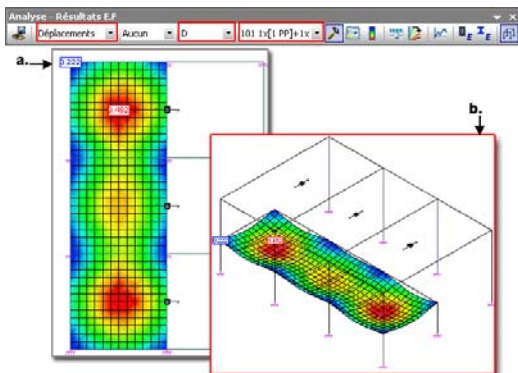
1. Right-click in the graphic area and deselect from context menu "Display nodes".
2. To view the displacement results on the storey slab:

- a. Define a top view of the workplane: use the shortcut **Alt + 3**.

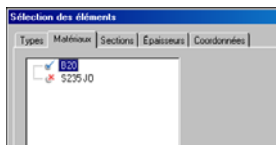
From the **Analysis - F.E. Results** toolbar, select the result type: **Displacements**, the planar elements results **D** and the case: combination no. **101**. Click on  icon to perform the exploitation. Access the results configuration dialog box (press **Alt + Z**), go to "Options" tab and check "Extreme values".

- b. Define a (-1, -1, 1) view of the workplane: click on  icon from **Predefined views** toolbar.

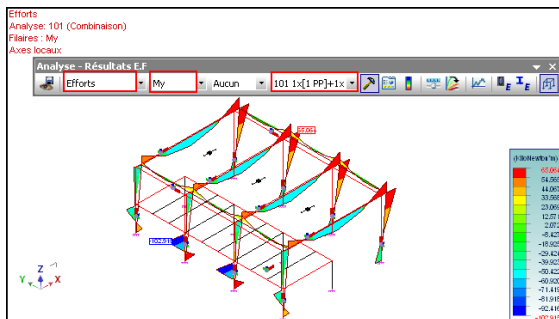
In the results configuration dialog box - "Options" tab: check "Display results on the deformed plot".



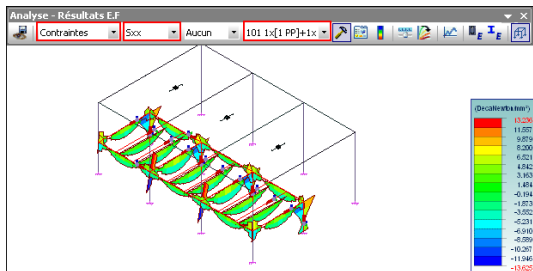
3. Right-click in the graphic area and deselect from context menu "Display the mesh".
4. To view forces results on the concrete linear elements:
 - a. Select the concrete elements using the selection by criteria: press **Alt + S** and, in the "Elements selection" dialog box, access "Materials" tab and select **B20**. Click "OK" to apply.



- b. In the **Analysis - F.E. Results** toolbar, select the result type: **Forces**, the linear elements results: **My** and the case: combination no. **101**. Click on icon to perform the exploitation.



5. To view stresses on steel elements:
- At first, right click in the drawing area and select from the context menu "Cancel selection".
 - Define a new selection by criteria: press **Alt + S** and, in the "Elements selection" dialog box, access "Materials" tab and select **S235JO**. Click "OK" to apply.
 - In the **Analysis - F.E. Results** toolbar, select the result type: **Stresses**, the linear elements results: **Sxx** and the case: combination no. **101**. Click on icon to perform the exploitation.




To clear the results displayed on the screen: keep the **Esc** key pressed for a few seconds.

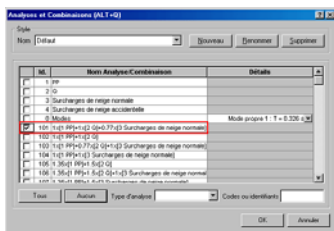
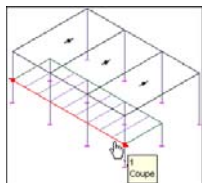
Result curves


The "Result curves" command available in the exploitation step of the project allows you to view different results (FE results as displacements, forces, stresses and concrete reinforcement area).

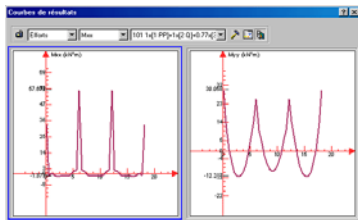
You can obtain result curves on the linear elements, and also on the planar elements with the help of section cuts. You can configure the result curves diagram using various options available from the diagram's window. You can also save the diagram as an image or print it using specific commands.

 **Example:** Display result curves on a section cut

1. To create the section cut: right-click in the drawing area and select from the context menu **Generate an entity > Generate a section cut**
2. Draw the section cut on the length of the storey slab, as shown
3. To select the analyses displayed on the curve: access the "Analyses and combinations" dialog box using **Alt + Q** shortcut. Select here only **101** combination case.



4. Select the section cut and click on  icon from the **Analysis - F.E. Results**: the default result curves (Mxx and Myy) for the selected analyze are automatically displayed:

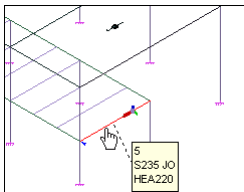


Stresses diagrams

The section stresses command allows the analysis of the stresses distribution on a given section. You will obtain a stresses diagram which displays dynamically the stresses results on each point of the concerned linear element.

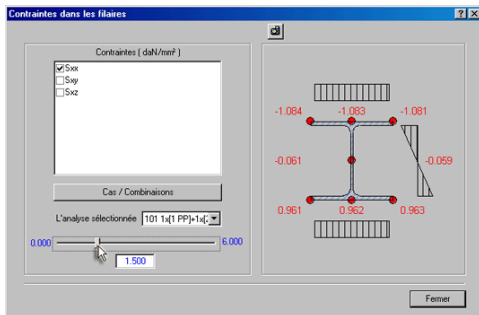
Example: Display a stresses diagram

1. Select the storey beam on which we will 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 (**S235JO** material, **HEA220** section) - click to select it.



2. Access the section stresses command: select from menu **Analyze > Section stresses...**

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






Post-processing animation

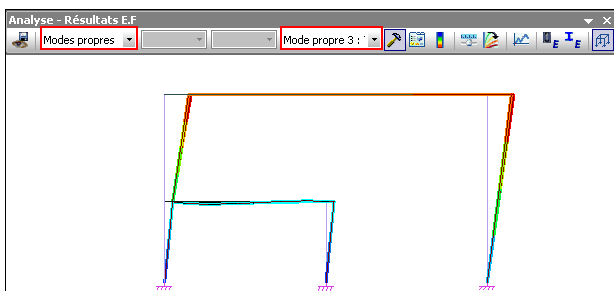
Advance Design allows you to create animation from the graphical results exploitation, following the results distribution and the deformed shape of the structure.


The **Animation** toolbar gives access to all necessary commands for creating and recording animation.



 **Example:** Create a postprocessing animation

1. From **Analysis - F.E. Results** toolbar, define the following results exploitation:
 - ✓ Select the "Eigen modes" result type
 - ✓ Choose "Eigen mode 3" from the analyses combo-box
 - ✓ Click on  icon to perform the exploitation
2. Define a front view of the workplane: click on  icon from **Predefined views** toolbar

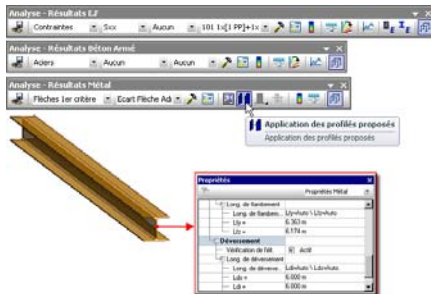


3. From **Analysis - F.E. Results** toolbar: click on  icon to view the exploitation results in animation.
To stop the animation: press **Esc** key.

Expert Design postprocessing

In the postprocessing phase, once the corresponding calculations are done, it is possible to view the results of the concrete / steel verifications. Moreover, you can optimize the concrete and steel members of the structure using the functions provided by these design modules.

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




Reinforced concrete results

Analysis - Reinforced concrete results toolbar, available after the concrete calculation, allows you to view the reinforcement results on concrete elements (reinforcement area; buckling lengths; reinforcement ratios).

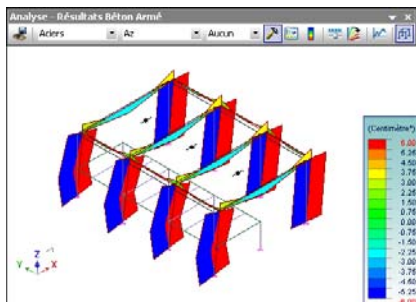


In the properties window of certain concrete elements (columns) it is possible to view the interaction curves issued from the reinforcement parameters, which are either determined automatically by the concrete module or imposed by the user. Therefore, it is possible to adjust, for example, a highly slender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


 **Example:** View longitudinal reinforcement on linear elements

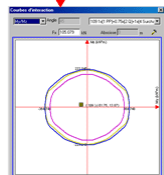
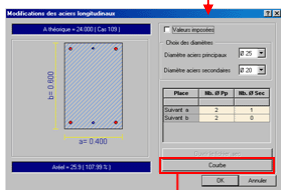
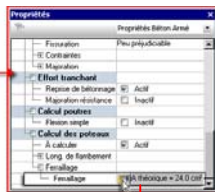
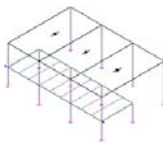
1. Define a (-1, -1, 1) view of the workplane, pressing **Alt + 6** keys
2. From **Analysis - Reinforced Concrete Results** toolbar:
 - ✓ Select the result type: **Reinforcement**
 - ✓ Select the result on linear elements: **Az**
 - ✓ Click on  icon to execute

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



 **Example:** View reinforcement results on a column

1. Select one column of the portal frame.
2. Access the element's properties window and go to **Expert Design** category - **Reinforcement** field.
3. Click on  icon to open the "Modification of longitudinal reinforcement" dialog box. You can view here the value of real reinforcement and also the calculated reinforcement for the selected column.
4. Click on "Curve" button to access the interaction curves window.



You can view here the position of the force component relative to the interaction area.

For advanced visualization options, double click on 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 allows you to verify deflections, the section's resistance, the element's stability according to second order effects (buckling and lateral-torsional buckling), and to optimize the steel shapes.

The steel results exploitation commands are available from the **Analysis - Steel Results** toolbar, once the steel calculation is done.



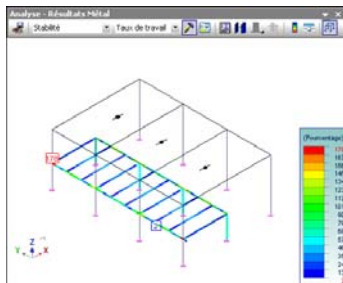
You can view the calculated buckling and lateral-torsional buckling parameters for each steel element in its properties window.

Example: Verify the steel elements stability

1. From **Analysis - Steel Results** toolbar:

- ✓ Select the result type: **Stability**
- ✓ Select the result on linear elements: **Work ratio**
- ✓ Click on icon to execute

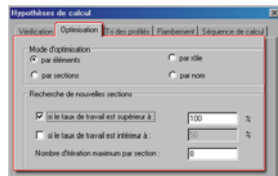
2. Access "Results" dialog box pressing **Alt + X** keys
3. Go to "Options" tab and select "Extreme values"
4. Press "OK" to execute



Steel elements optimization


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

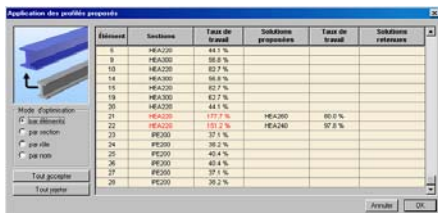
The program identifies the steel shapes with a higher / lower work ratio than specified and proposes more appropriate sections.



You can accept globally or partially the proposed shapes then rerun the FE calculation and the structure optimization. You can iterate these operations until you obtain the appropriate work ratio for all steel shapes.

Example: Optimization of steel shapes

1. From **Analysis - Steel Results** toolbar: click on  icon. The "Proposed shapes" dialog box opens; the steel sections with a work ratio out of the specified range are displayed in red.



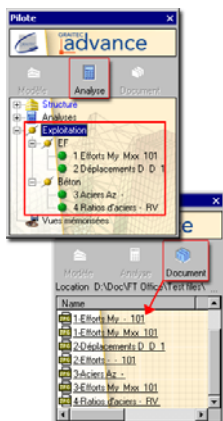
Epaisseur	Section	Taux de travail	Solutions proposées	Taux de travail	Solutions retenues
6	HEA200	44.1 %			
8	HEA300	58.8 %			
10	HEA200	62.7 %			
14	HEA300	66.6 %			
15	HEA200	62.7 %			
19	HEA300	62.7 %			
30	HEA300	44.1 %			
21	HEA200	177.7 %	HEA200	88.8 %	
22	HEA200	191.2 %	HEA200	95.6 %	
23	IP200	37.1 %			
24	IP200	38.2 %			
25	IP200	45.4 %			
26	IP200	40.4 %			
27	IP200	37.1 %			
29	IP200	38.2 %			

2. Press "Accept all" button to accept all proposed shapes
3. Press "OK" to close and apply
4. Run again the steel calculation: select from menu **Analyze > Steel calculation**
5. After calculation, open again the "Proposed shapes" dialog box. If there are other proposed shapes, repeat the above steps until all shapes are working at the specified ratio.

Memorized exploitations

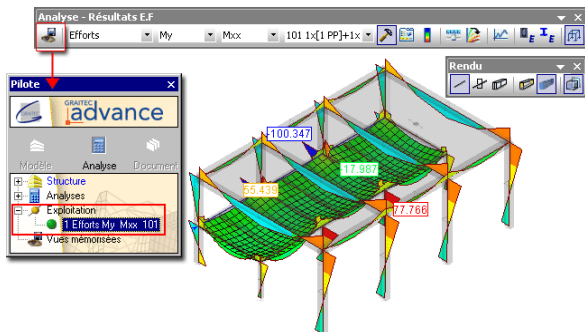
An exploitation view memorizes the whole exploitation scenario (result type, result component, selected analyses and elements, results visualization settings), together with the display settings of the model (view point, rendering...). For each exploitation view, a correspondent image file is saved on the disc. You can find the saved images in the **Document** mode of the pilot.

Exploitation views help you automatically replay anytime the memorized exploitations, without having to manually recreate the exploitation scenario. Moreover, if you have changed the structure hypotheses and the results have been modified, the updated exploitation views display the new results.



Example: Create an exploitation view

1. Define a (1, -1, 1) view of the workplane, pressing **Alt + 5** keys
2. From **Rendering** toolbar: click on icon to set a ghost display of the descriptive model
3. Access "Results settings" command pressing **Alt + Z** keys; in "F.E." tab: select the result type: **Forces**, the result on linear elements: **My** and the result on planar elements: **Mxx**
4. From **Analysis - F.E. Results** toolbar: click on icon to memorize the exploitation



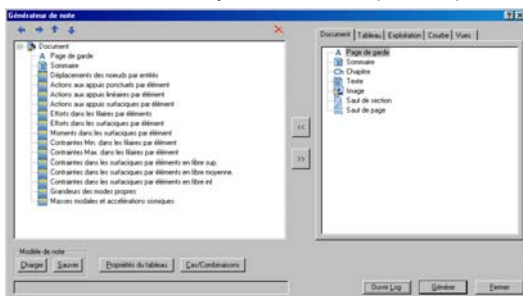
5. To replay the memorized exploitation: double click on it in the pilot


Reports

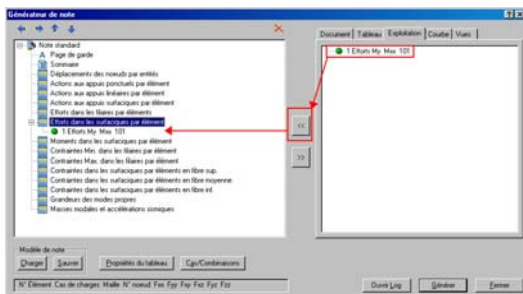
Advance Design provides an advanced and powerful report generator tool, which allows you to easily define the desires reports. You can use the available report templates, or you can define your own template. The report generator filters its content function of the current hypotheses and available results. At the same time, the report content takes into account the selection of elements (if any).

 **Example:** Generate a report

1. Choose from menu **Documents > Standard report...**: the report generator loads automatically the standard report template.



2. Go to "Exploitation" tab of the report generator, and select the exploitation available there.
3. In the report's content section, select the table with forces on planar elements and click on  button to insert the exploitation right below.



4. Click on "Generate" button to start the report creation. When finished, the report is automatically displayed with the document viewer application.

France

GRAITEC France Sarl

10bis Burospace
91572 Bièvres

Tel. 33 (0)1 69 85 56 22
Fax 33 (0)1 69 85 33 70
Web <http://www.graitec.com/Fr/>
Email info.france@graitec.com

Canada

CivilDesign Inc

183, St. Charles St. W. Suite 300
Longueuil (Québec) J4H 1C8

Tel. (450) 674-0657
Fax (450) 674-0665
Hotline (450) 674-0657 (VisualDesign)
Web <http://www.civild.com/>
Email sales@civild.com

Germany, Switzerland, Austria

GRAITEC GmbH

Centroallee 263a

D-46047 Oberhausen Germany
Tel. +49-(0) 208 / 62188-0
Fax +49-(0) 208 / 62188-29
Web <http://www.graitec.com/Ge/>
Email info.germany@graitec.com

Romania

GRAITEC Roumanie SRL

Str. Samuil Vulcan, Nr. 10 Sector 5
București, Romania

Tel. +40 (21) 410 0119
Fax +40 (21) 410 0124
Web <http://www.graitec.com/Ro/>
Email info.romania@graitec.com

USA

GRAITEC Inc.

Dallas / Forth Worth

Tel. (877) 464-3366
Fax (450) 628 0400
Hotline (877) 464-5046
Web <http://www.graitec.com/En/>
Email info.usa@graitec.com

Canada

GRAITEC Inc.

49 Rue de la Pointe-Langlois
Laval (Québec) H7L 3J4

Tel. (877) 464-3366
Fax (450) 628 0400
Hotline (877) 464-5046
Web <http://www.graitec.com/CaFr/>
Email info.canada@graitec.com

Czech Republic and Slovakia

AB Studio spol. s r.o.

Jeremenkova 90a 140 00 PRAHA 4

Tel. +420/244 016 055
Fax +420/244 016 088
Hotline +420/244 016 050
Web <http://www.abstudio.cz/>
Email abstudio@abstudio.cz

United Kingdom

Adris Limited

Riverside House, Brunel Road,
Totton, Southampton, Hampshire,
SO40 3WX. England

Tel. +44 023 8086 8947
Fax 44 023 8086 1618
Hotline +44 023 8086 9995
Web <http://www.adris.co.uk/>
Email sales@adris.co.uk