ADVANCE DESIGN

TUTORIAL

GRAITEC

www.graitec.com

Advance Design

Tutorial

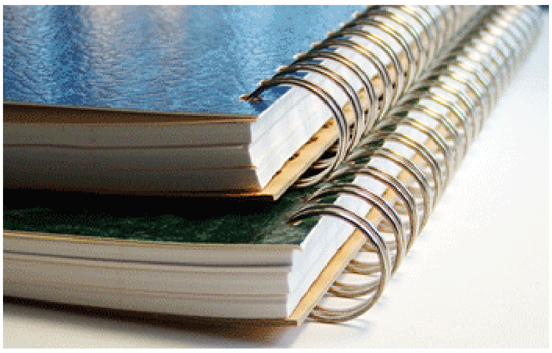


Table of Contents

About this tutorial

This tutorial contains step-by-step instructions for creating and modeling structure elements, setting the structure assumptions, calculating and optimizing the structure and generating reports of the results.

In the first part of this tutorial you will follow the steps for creating and configuring the settings of an Advanced Design project.

In Lesson 2 you will create and model a structure using various CAD tools (e.g., workplane, coordinate systems, snap modes etc.) and CAD functions (e.g., copy, move, rotate, subdivide, trim, extend, create symmetries etc.).

In the following 6 lessons you will input the structure assumptions: configure loadings and analysis types.

In Lesson 9 you will create the mesh using the mesh engines (Advanced and Standard Mesh) and calculate the structure using a new generation solver engine.

In Lesson 10 you will view the results by selecting options from a large set of visualization options. You will also save the post-processing results.

In Lesson 11 you will generate calculation reports using a variety of predefined result tables and insert post-processing views.

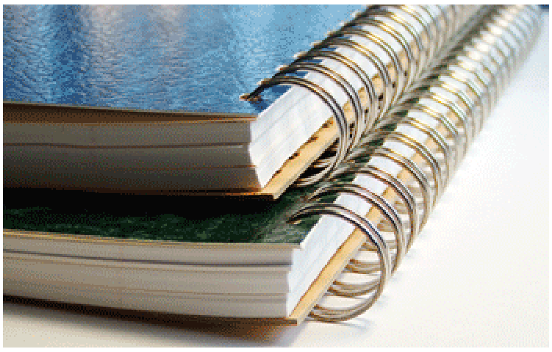
In the next 2 lessons you will calculate and optimize the reinforced concrete and steel structure according to selected standards.

In Lesson 14 you will use the column reinforcement analysis tool to check the calculated reinforcement area for a column aided by the interaction curves.

In the final lesson you will generate a post-processing animation.

In this tutorial:

- *Lesson 1: Preparing and organizing your model*
- *Lesson 2: Modeling and managing your structure*
- *Lesson 3: Creating windwalls and defining snow and wind supporting elements*
- *Lesson 4: Defining Self Weight Loads and Live Loads*
- *Lesson 5: Automatic creation of wind loads*
- *Lesson 6: Automatic creation of snow loads*
- *Lesson 7: Defining the modal and seismic analysis*
- *Lesson 8: Automatic creation of load combinations*
- *Lesson 9: Meshing and FE Calculation*
- *Lesson 10: Post-processing finite elements results*
- *Lesson 11: Creating reports and post-processing views*
- *Lesson 12: Steel design and shape optimization*
- *Lesson 13: Reinforced concrete design*
- *Lesson 14: Column reinforcement analysis*
- *Lesson 15: Creating a post-processing animation*



How to use this guide

This tutorial offers a description of the main functions of Advance Design, and the program's working process. It does not describe all Advance Design features and commands. For detailed information about all commands and functions, access the Online Help from the program.

Note: *The model presented in this tutorial is for learning purpose and does not conform to specific standards.*

Lesson 1: Preparing and organizing your model

This lesson describes how to prepare and organize a new project.

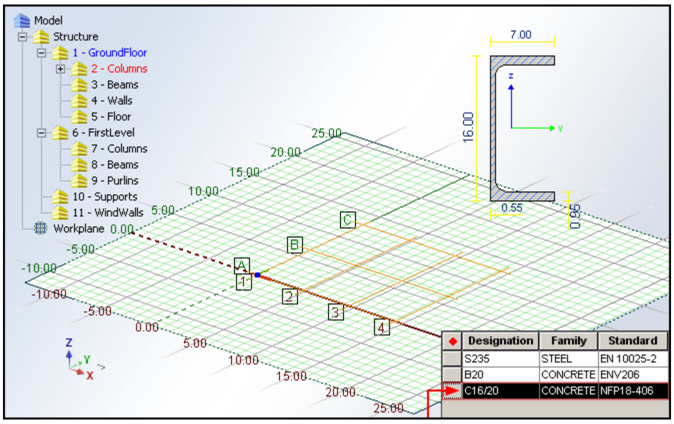


Figure 1: Systems and subsystems of the structure, model configuration

You will learn how to:

- Start a new project.
- Define the materials.
- Define the cross sections.
- Create systems and subsystems for easier model handling.
- Define the view on the workplane.
- Define a grid.

Step 1: Start Advance Design

From the Windows **Start** menu, select **Programs > Graitec > Advance Design > Advance Design**. The program starts and the Start Page appears.

***Note**: To prevent the display of the start page each time Advance Design is launched, select the **Do not show Start Page next time** option.*

Step 2: Configure settings for a new project

1. In the start page, **GRAITEC Resources** area, click **Configuration**.
2. In the "Localization configuration" dialog box, make the following settings:

   In the "Localization" area:

   – From the "Language" drop-down list, select the interface language: **English**.

   – From the "Reports Language" drop-down list, select **English**.

   In the "Standards" area select the standards to use in the Advance Design project:

   – Seismic: **EC8**.

   – Climatic: **EC1**.

   – Reinforced Concrete: **EC2**.

   – Steelwork: **EC3**.

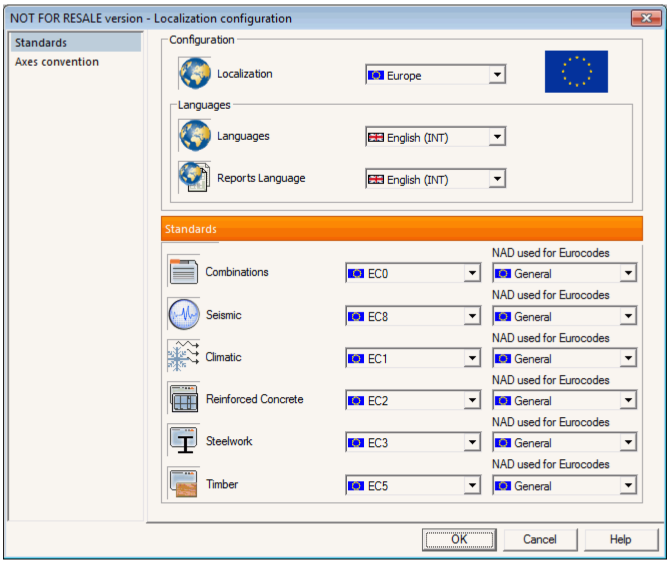


Figure 2: The "Localization configuration" dialog box

3. Click **OK** to apply and close the dialog box.

Step 3: Create a project

In this step you will create an Advance Design project and configure its parameters.

1. In the start page, **My Projects** area, click **New**.
2. In the "Project settings" dialog box, enter the data for the project description: the project's name, set, address, reference number, progress stage and date of creation. These details appear in the reports you may generate from the current project.
4. Click **Next** to continue.
5. In the next window, enter the settings for the workspace, the characteristics (the reference temperature and the default material) and the units to use in the project.

In this tutorial the following settings are used:

– 3D workspace.
– Bending rigid structure option enabled.
– Front view as default view.
– 0.00 C reference temperature.

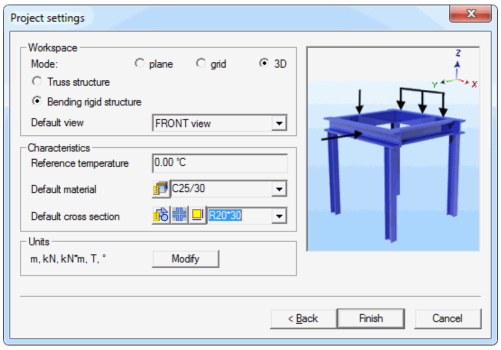


Figure 3: "Project settings" dialog box

The working units:

– Lengths: Meter.
– Forces: KiloNewton.
– Moments: KiloNewton*m.
– Stresses: MPa (N/mm^2).
– Displacements: Centimeters.
– Cross sections dimensions: Centimeters.

***Note**: The **Modify** button displays the "Working units definition" dialog box, where the working units type and precision can be configured.*

6. Click **Finish** and close the dialog box.

7. From the main menu, select **File > Save as**. In the "Save As" dialog box, "File name" field, enter **Tutorial**.

8. Click **Save** and close the dialog box.

Step 4: Define materials

In this step you will add the **CONCRETE C16/20** material type to the list of materials in the model.

1. In the Pilot, right click **Model** and select **Used materials** from the context menu.

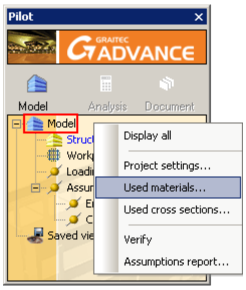


Figure 4: The "Model" context menu

The "Materials" dialog box appears. It contains the default list of materials available for the current model.

2. Click **Libraries** to display the “Libraries” panel.
3. From the “Family” drop-down list, select **CONCRETE**.
4. From the “Standard” drop-down list, select the material standard: **EN206**.
5. Select **C16/20** material.
6. Click **Import** to add it in the list of materials available for the current model.

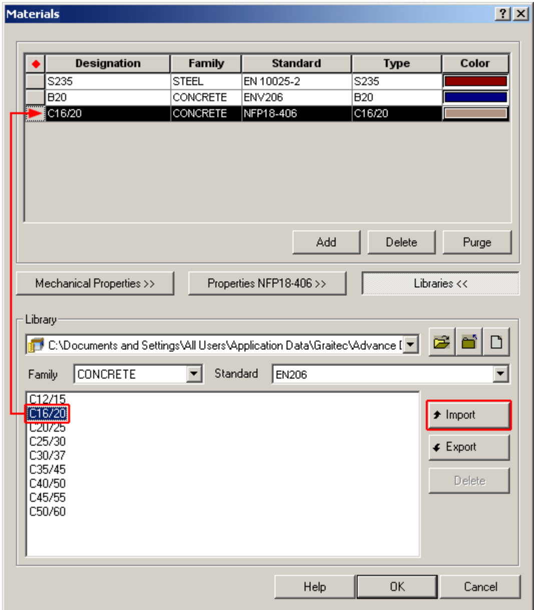


Figure 5: Adding a new material from a library

7. Click **OK** to apply and close the dialog box.

Step 5: Define cross sections

Next, define the cross sections to use in the model: new defined parametric cross sections (**C40, R30*55** and **R20*35**) and a cross section from the libraries (**UPE160**).

Adding cross sections using defined parameters

1. In the Pilot, right click **Model** and select **Used cross sections** from the context menu.

   The “Description of defined geometries” dialog box appears.

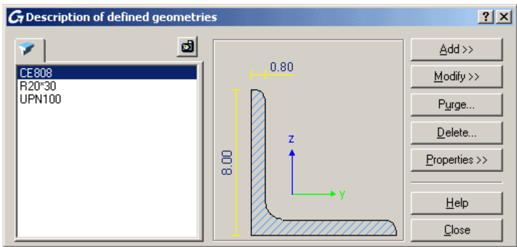


Figure 6: “Description of defined geometries” dialog box

   The panel on the left side of the dialog box displays the list of default cross sections available for this model.

2. Click **Add**. The dialog box is extended with four tabs: **Defined**, **Libraries**, **User** and **Compound cross sections**.
3. Select the **Defined** tab to add new cross sections using defined parameters:
   - From the “Type” drop-down list, select **Square**.
   - In the table located below, change the **Width** value to **40**.

4. On the **Defined** tab, click **Add** to import the new cross section in the current list.

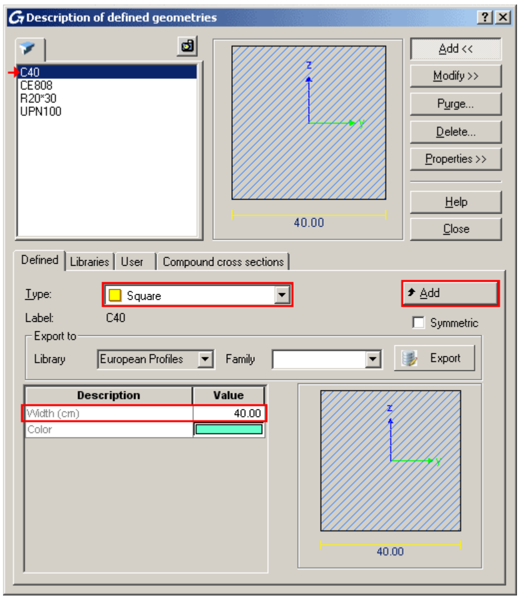


Figure 7: Adding a square cross-section in the current list

Add two rectangular cross sections: from the “Type” drop-down list, select **Rectangular** and define:

– The first rectangular cross section with a height of **55 cm** and a width of **30 cm**.

– The second rectangular cross section with a height of **35 cm** and a width of **20 cm**.

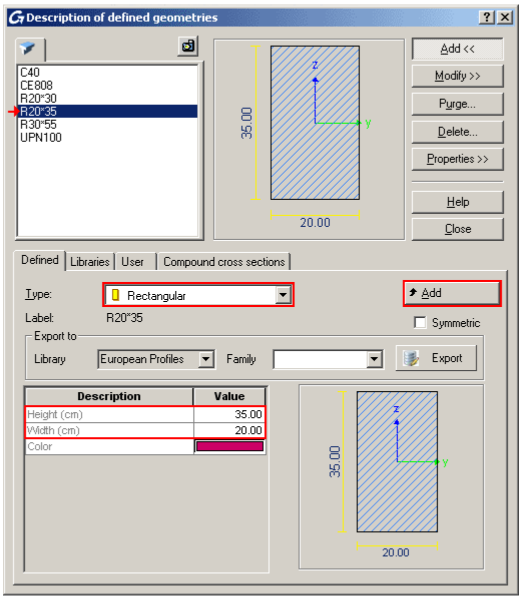


Figure 8: Adding a rectangular **35*20** cross section in the current list

Adding a cross section from the available libraries

1. In the “Description of defined geometries” dialog box, select the **Libraries** tab.
2. Expand the **European Profiles** tree structure.
3. Scroll down and select the **UPE** cross sections family.
4. From the table below, select the **UPE160** cross section type.
5. Click **Import** to add it in the current list.

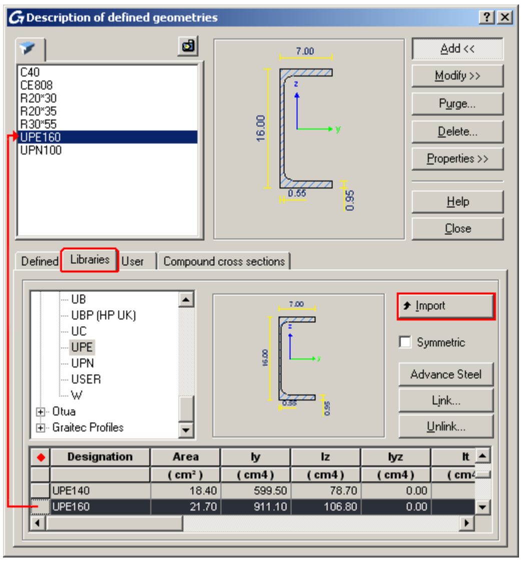


Figure 9: Adding a cross section from the Library

The selected cross section shape is displayed in the preview area of the dialog box.

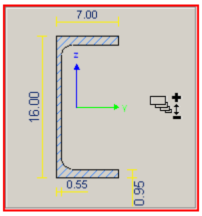


Figure 10: Cross section preview in the “Description of defined geometries” dialog box

In the preview area, keep the left mouse button pressed and drag to move the image. Right click and drag to zoom the image. Drag down for zoom out and up for zoom in.

6. Click **Close**.

Step 6: Create systems and subsystems

This step describes how to organize the structure elements by systems and subsystems, and how to manage the system tree. You will create the structure's systems and subsystems and you will define their properties.

Creating a system

1. In the Pilot, right click **Structure** and select **System management > Create a subsystem** from the context menu.

   A new system is created in **Structure**.

2. Enter the name of the new system: **GroundFloor**.

Setting the system properties

1. In the Pilot, click the **GroundFloor** system to select it.
2. Define the system attributes in the Properties window:

If the Properties window is hidden, hover the mouse cursor over the Properties tab, docked on the right margin of the workspace. Click the auto hide button to permanently display the properties window.

<u>Level</u>

– Select the **Level** option to enable the work by levels function for the selected system. Thus, you can create structure elements according to specified coordinates.

– Enter **3.20 m** in the “top Z” field to define the level height.

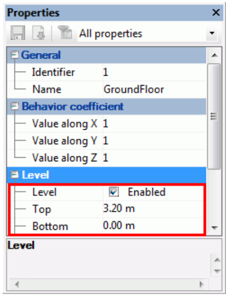


Figure 11: The system “Properties” window

Create the following systems and subsystems:

– In the Pilot, right click **GroundFloor** and select **System management > Create a subsystem**. Name the new subsystem **Columns**.

*To create a subsystem faster, select the system where you want to create it and press <**F6**>.*

– Select the **GroundFloor** system and press <**F6**> to create the **Beams** subsystem. Using the same process, create the **Walls** and **Floor** subsystems in the **GroundFloor** system.

– Select **Structure** and create a system named **FirstLevel**.

– Select the **FirstLevel** system and create the **Columns**, **Beams** and **Purlins** subsystems.

– Create two new systems in **Structure**: **Supports** and **WindWalls**.

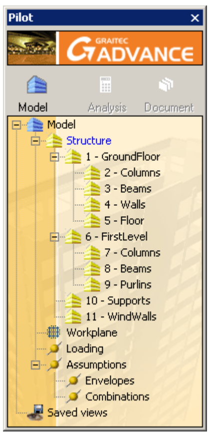


Figure 12: Systems tree structure in the Pilot

*To rename a system faster, select it and press <**F2**>.*

Step 7: Define the workplane

In this step, you will define a workplane by projection on the active view and use the predefined views.

Press <**ALT + R**> and drag to rotate the view.

Notice the 3D axes orientation:

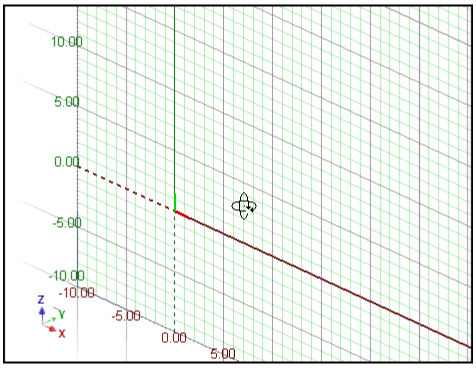


Figure 13: 3D view on the workplane

1. On the **Predefined views** toolbar, click  for a top view on the workplane.

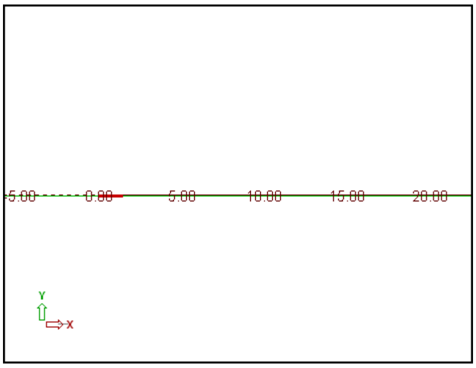
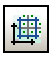


Figure 14: Top view on the workplane

2. On the **Workplane** toolbar, click  to define a workplane by projection on the active view.

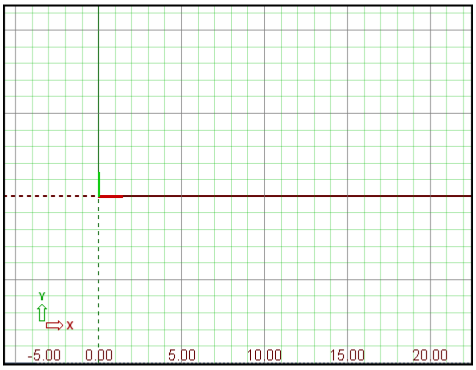


Figure 15: The workplane defined on the **XY** plane

Step 8: Define the grid

The grid is an intersecting lines system that helps to easily input structure elements relative to certain coordinates. You will create a grid with **3** blocks on the **X**-axis and **2** blocks on the **Y**-axis.

Press <**ALT + R**> and rotate for a 3D view.

1. In the Pilot, select the **GroundFloor** system.
2. From the main menu select **Generate > Grid**.
3. Create the grid by snapping to the workplane:
   - Click in the workspace on the (**0, 0**) coordinates point to define the grid's origin.

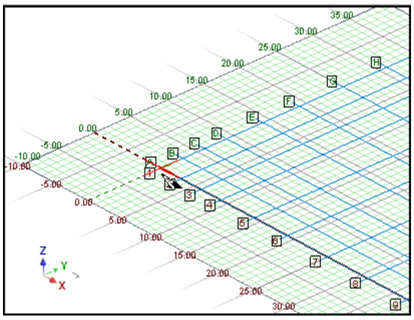


Figure 16: Defining the origin of the grid

   - Click in the +**X** direction to define the **X**-axis of the grid.

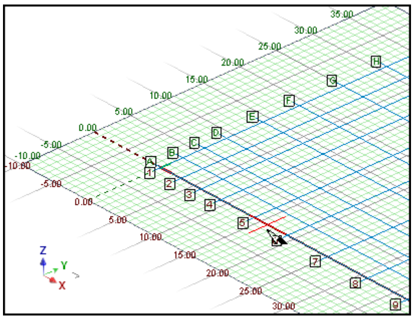


Figure 17: Defining the X direction of the grid

   - Click in the +**Y** direction to define the **Y**-axis of the grid.

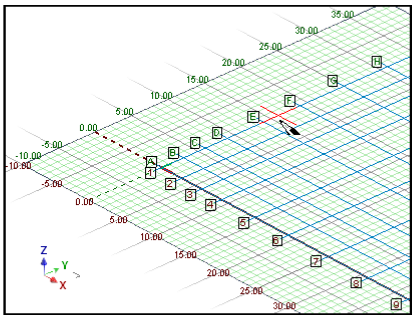


Figure 18: Defining the **Y** direction of the grid

4. Press <**Esc**> to disable the grid tool.

Modifying the grid properties

1. Click the grid to select it.

2. In the Properties window, modify the grid parameters:

X Options

– In the "Definition" field, enter **3*4.75** (i.e., **3** blocks of **4.75** meters each).

Y Options

– In the "Definition" field, enter **2*6.50** (i.e., **2** blocks of **6.50** meters each).

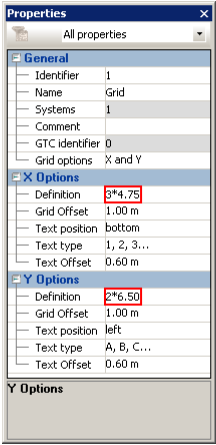


Figure 19: Grid Properties window

Hide the workplane for a clear view of the user-defined grid: in the Pilot, right click **Workplane** and select **Hide**.

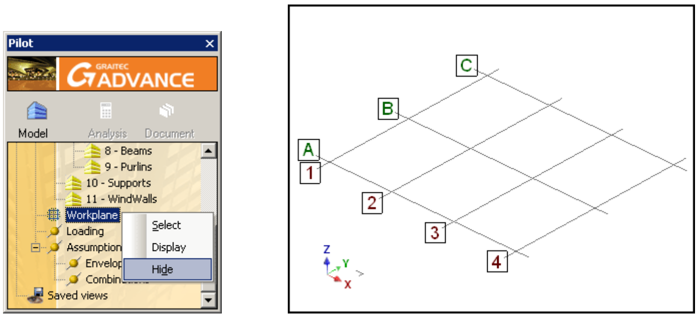


Figure 20: Hiding the Workplane

Double click any element or system of elements in the Pilot to hide / display them in the work area. When hidden, the corresponding icon from the pilot appears grey.

Select the grid and drag and drop it on the **Columns** system.

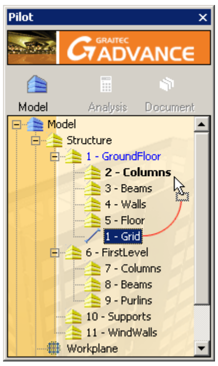


Figure 21: Placing the user-defined grid in the **Columns** subsystem

Structural elements can be placed in systems by drag and drop, in the Pilot.

Lesson 2: Modeling and managing your structure

In this lesson you will create and adjust the elements of a structure in one level using CAD commands available in Advance Design: copy (by mirror and by rotation), move, split and extend. The ground floor of the structure consists of concrete columns with rigid point supports, concrete walls with rigid linear supports and concrete beams. The first level has a concrete floor and concrete columns. The top of the first level of the structure has concrete beams and steel purlins.

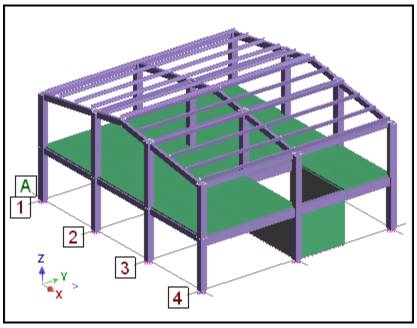


Figure 22: The descriptive model of the structure

You will learn how to:

- Create and define structural elements: columns, beams, purlins, walls and floor.
- Use the CAD functions: copy, move, split, extend and mirror.
- Use display settings for easier model handling.

Step 1: Create, copy and modify structural elements

In this step you will create and modify structural elements using CAD tools.

Creating columns on the ground floor

1. In the Pilot, select the **Columns** subsystem of **GroundFloor** to define it as current.

Note: *The structure elements are automatically placed in the current system. Define a system as current by selecting it in the Pilot.*

The **GroundFloor** system coordinates were already defined (**Z top = 3.20 m, Z bottom = 0.00 m**). Thus, the columns of this level can be created by a single click, snapping automatically to the specified coordinates.

2. On the **Modeling** toolbar, on the **Create a linear element** flyout, click  to create a vertical linear element by 1 point.

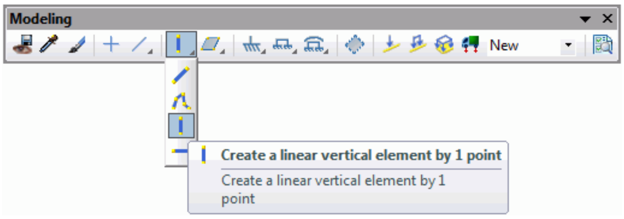


Figure 23: **Modeling** toolbar, create a linear element flyout - creating a linear vertical element by 1 point

3. In the Properties window, make the following settings:

Material

– Select the **C 16/20** material code.

Cross section

– From the "Extremity 1" drop-down list select the **C40** cross section.

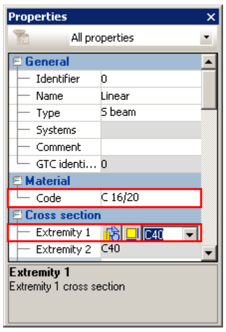


Figure 24: Selecting the linear element's material and cross section

4. Create 12 columns by snapping to each grid line intersection.

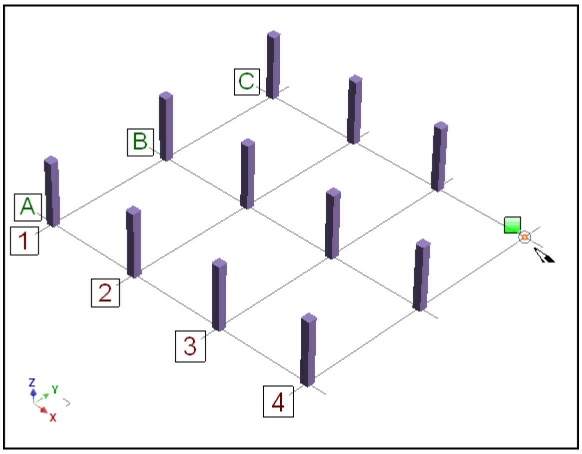
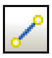


Figure 25: Creating the columns by snapping to grid line intersections

*Note: Make sure the **Snap modes** option is enabled (when enabled, it appears highlighted on the toolbar -* *).*

*On the **Snap modes** toolbar, click*  *to enable the endpoint snap.*

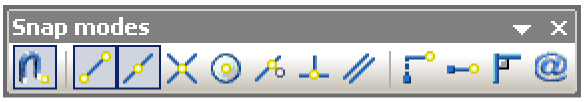


Figure 26: **Snap modes** toolbar

5. Press <**Esc**> to quit the drawing function.

*Note: After creating elements, press <**Esc**> or right click anywhere in the work area and select **Finish** from the context menu to quit the drawing function.*

Creating transverse beams

1. In the Pilot, select the **Beams** subsystem.
2. On the **Modeling** toolbar, on the **Create a linear element** flyout, click .

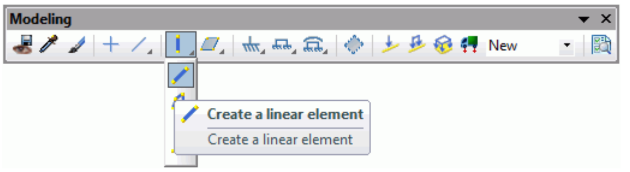


Figure 27: **Modeling** toolbar, create a linear element flyout - creating a linear element

3. In the Properties window, make the following settings:

<u>Material</u>

– Select the **C 16/20** material code.

<u>Cross section</u>

– From the "Extremity 1" drop-down list select the **R30*55** cross section.

– Expand "Eccentricity", click **Option** and from the drop-down list, select **(0, z-)** to define the position of the cross section relative to its own axis.

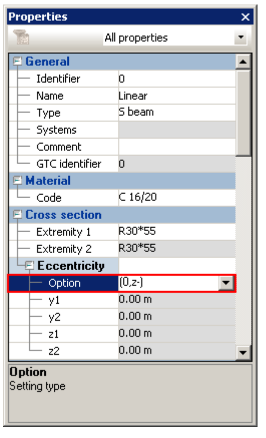


Figure 28: Defining the linear element eccentricity

Press <**ALT + 5**> for a 3D view.

4. Create a beam connecting the top extremities of the **A4** and **B4** columns.

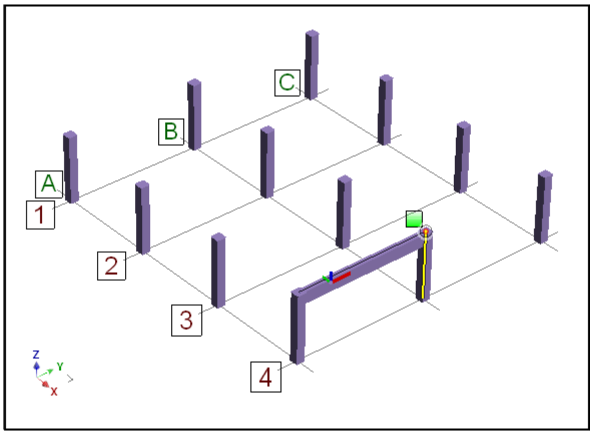


Figure 29: Creating a transverse beam

5. Create a second beam, connecting **B4** and **C4** columns.

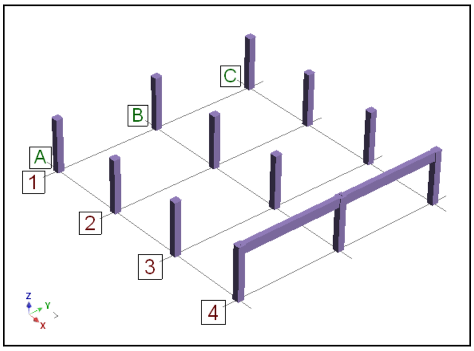


Figure 30: Creating the second beam

6. Press <**Esc**> to quit the drawing function.

Multiplying the transverse beams

Next, create several transverse beams using the copy by translation command.

1. Click the two beams to select them.

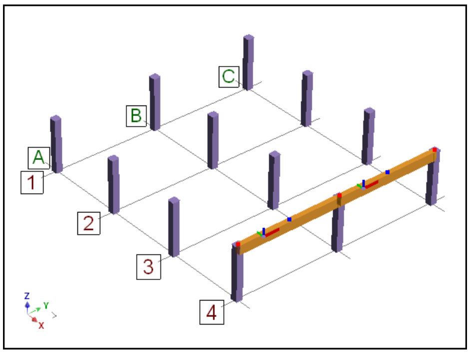


Figure 31: Selecting the beams

2. On the **CAD Modifications** toolbar, click .

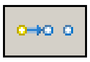
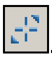
3. In the “Multiple copy” dialog box select the **Translation** option.

---

*Make sure the **Rotation** option is not selected.*

---

4. Define the translation parameters:

- In the “Number” field, input **3** to generate three copies.
- In the “Mode” category, click  to create copies of the selected objects at each defined interval.
- In the “Vector” category, click .

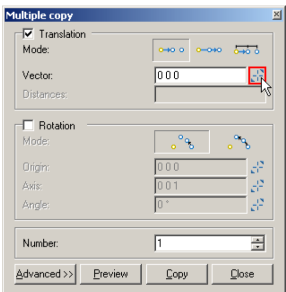


Figure 32: “Multiple copy” dialog box

– Specify the copy vector by snapping to the top extremities of the **C4** and **C3** columns.

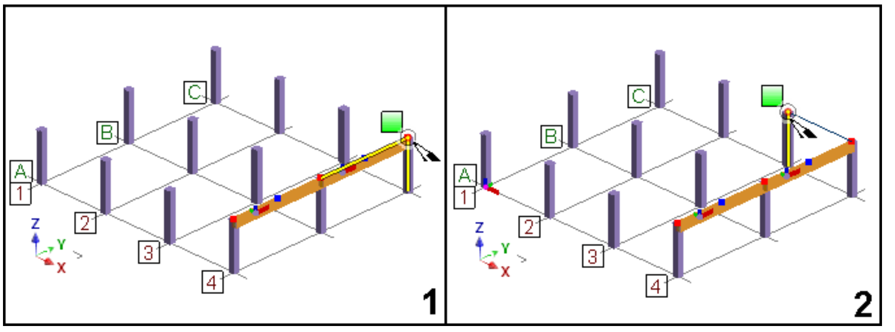


Figure 33: Defining the transverse beams copy vector

5. Click **Preview** to see how the settings are applied.

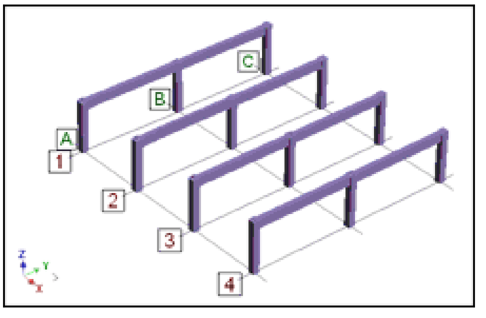


Figure 34: Transverse beams

6. Click **Copy** and close the dialog box.

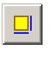
7. Press <**Esc**> to unselect the beams.

Modifying the properties of a transverse beam

1. In the Pilot, select linear element 15.

2. In the Properties window, make the following modification:

Cross section

– Click “Extremity 1” cell, then click  to open the “Defined” dialog box.

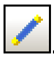
– In the **Height** field enter **80**.

– In the **Width** field enter **40**.

3. Click **OK**.

*Save your model regularly accessing **File > Save** from the main menu or pressing the <**Ctrl + S**> keys. To configure the automatic save: from the main menu select **Options > Application**. In the "Option - Application" dialog box, go to the **Folders** tab and input a value for the automatic save frequency (in minutes).*

Creating longitudinal beams

1. On the **Modeling** toolbar, on the **Create a linear element** flyout, click .

2. In the Properties window, make the following settings:

<u>Material</u>

– Select the **C 16/20** material code.

<u>Cross section</u>

– From the “Extremity 1” drop-down list select the **R20*35** cross section.

3. Create a beam connecting the top extremities of the **A1** and **A2** columns.

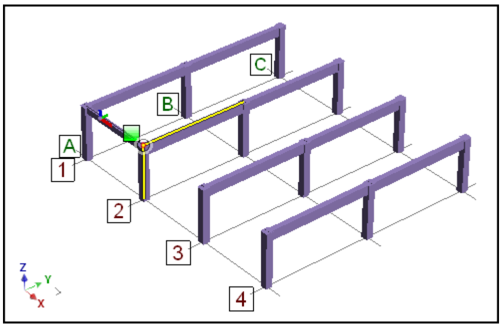


Figure 35: Creating a longitudinal beam

4. Create two other beams: the first one connecting the **A2** and **A3** columns and the second one connecting the **A3** and **A4** columns.

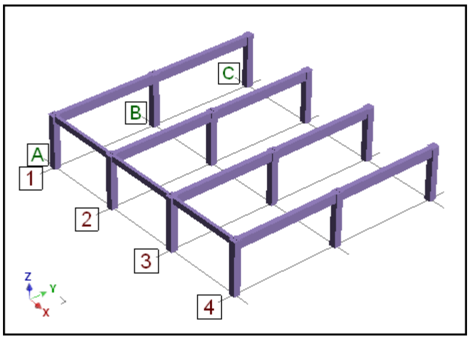


Figure 36: Creating longitudinal beams

5. Press <**Esc**> to quit the drawing function.

Multiplying the longitudinal beams

Next, create several longitudinal beams using the copy by translation command.

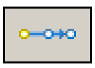
1. Select the three longitudinal beams.

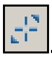
2. On the **CAD Modifications** toolbar, click .

3. In the “Multiple copy” dialog box, select the **Translation** option.

4. Define the copy by translation parameters:

– In the “Number” field, input **8** to generate eight copies.

– In the “Mode” category, click  (translation defined by a total vector divided in n copies) to create copies of the selected elements at even distances between them, within a specified interval.

– In the “Vector” category, click . Specify graphically the copy vector by snapping to the top extremities of the **A1** and **C1** columns, as in the following sequence of images:

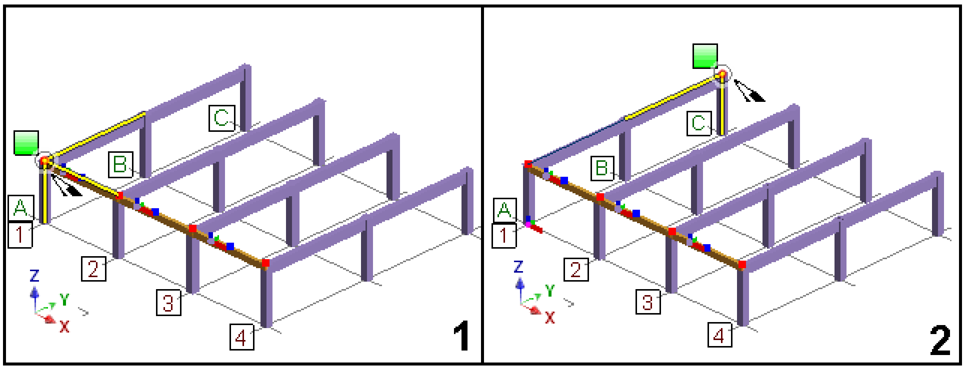


Figure 37: Defining the longitudinal beams copy vector

5. Click **Preview** to see how the settings are applied.

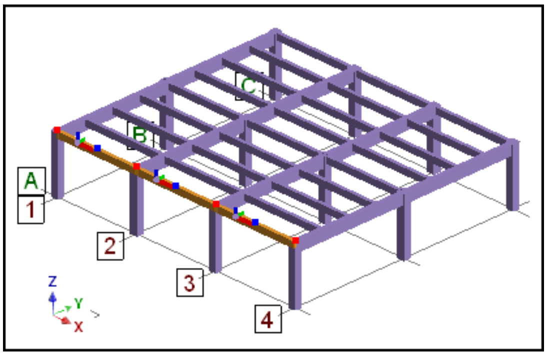
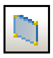


Figure 38: The longitudinal beams

6. Click **Copy** and close the dialog box.
7. Press <**Esc**> to unselect the beams.

Creating walls

Since the **GroundFloor** system coordinates are defined (**Z top = 3.20 m, Z bottom = 0.00 m**), it is possible to create walls by two points, snapping automatically to the specified coordinates.

1. In the Pilot, select the **Walls** subsystem.
2. On the **Modeling** toolbar, on the **Create a planar element** flyout, click  to create a vertical planar element by 2 points.

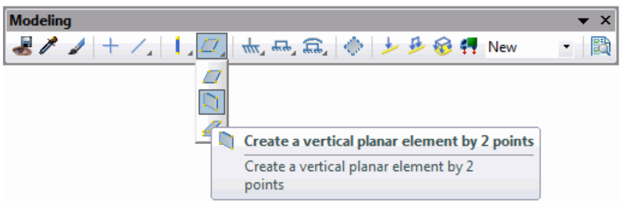


Figure 39: **Modeling** toolbar, create a planar element flyout – creating a vertical planar element by 2 points

3. In the Properties window, make the following settings:

<u>Material</u>

– Select the **C 16/20** material code.

<u>Thickness</u>

– In the “1st vertex” field, enter **15 cm**.

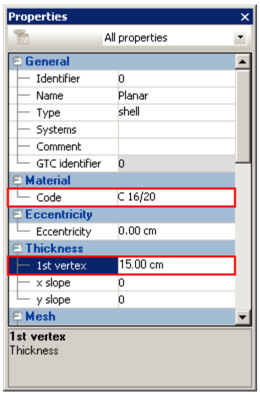


Figure 40: Setting the wall thickness

4. Create a wall connecting the **B3** and **B4** columns.

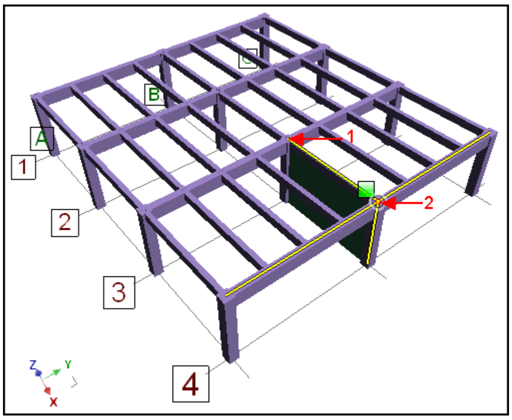


Figure 41: Creating a wall on the ground floor

With the **Create a vertical planar element by 2 points** tool still enabled, create two other walls (Figure 42).

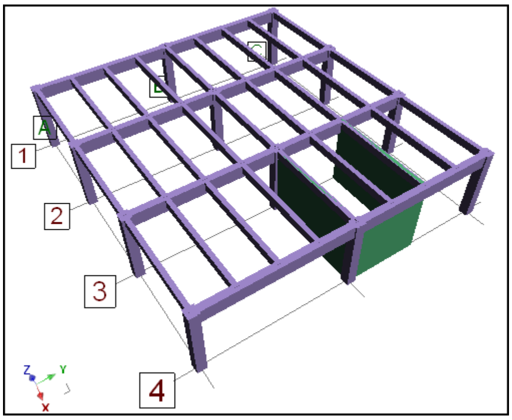
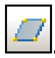


Figure 42: The walls of "GroundFloor”

Creating a floor

1. In the Pilot, select the **Floor** subsystem.

2. On the **Modeling** toolbar, on the **Create a planar element** flyout, click .

3. In the Properties window, select the **C16/20** material code.

4. Create a floor as in the next sequence of images:

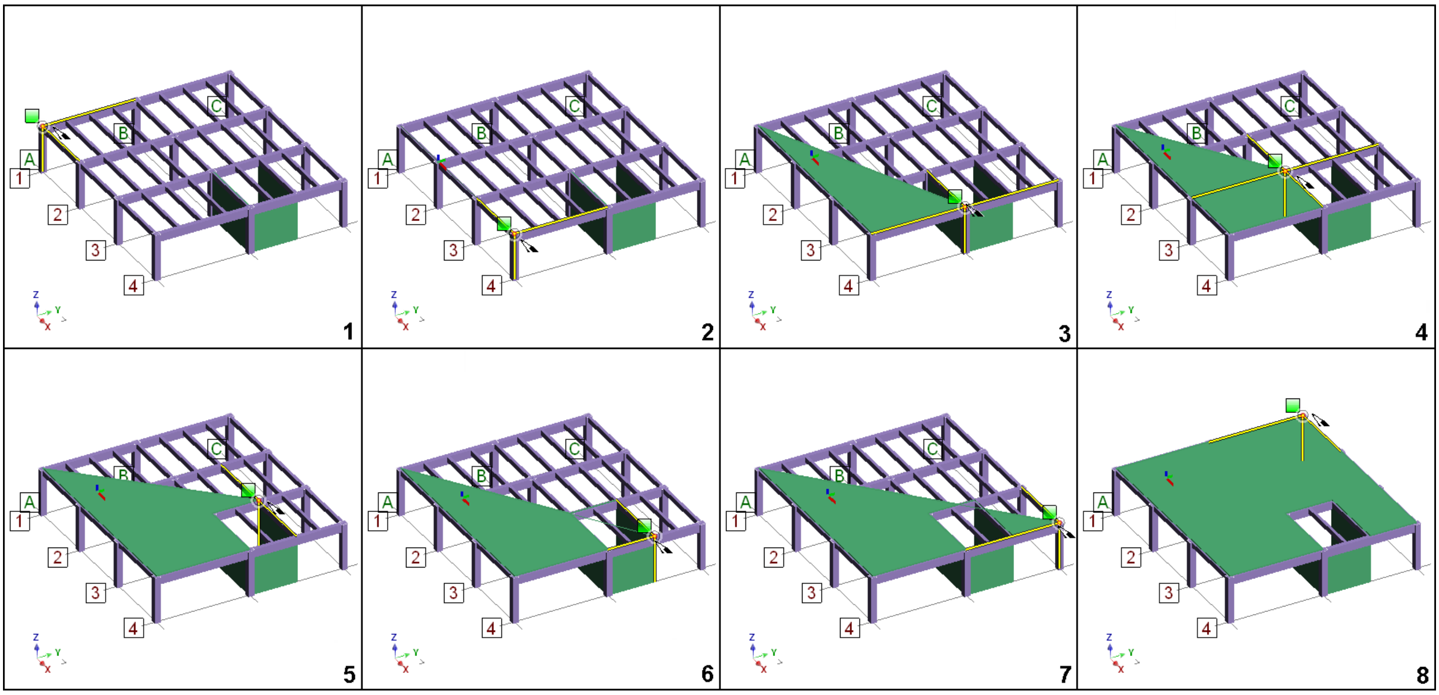


Figure 43: Creating the floor of the first level

5. Press <**ENTER**> to finish.

Press <**Esc**> to quit the drawing function.

Delete the unnecessary beams: select the three beams as shown (Figure 44) and press <**DELETE**>.

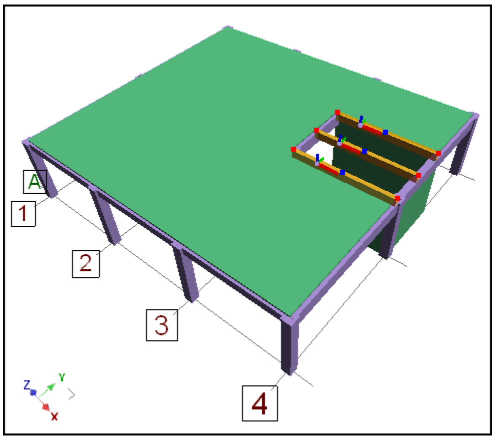


Figure 44: The beams to delete

***Note**: To select a certain element from an area where several elements overlap, click in the area of the element to select and press <**TAB**> until the desired element is highlighted.*

Creating columns on FirstLevel

Next, copy columns of the ground level to the **FirstLevel** system.

1. Select the columns along the **1**, **4** and **A** axes (Figure 45).

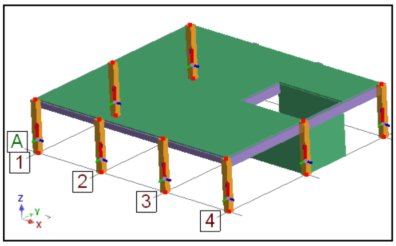


Figure 45: Selection of the columns

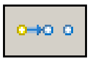
*Use <**ALT + R**> and drag to rotate around the model. Press <**Esc**> to disable the rotating tool.*

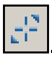
2. On the **CAD Modifications** toolbar, click .

3. In the “Multiple copy” dialog box, select the **Translation** option.

4. Set the translation copy parameters:

– In the “Number” field, input **1** to generate a copy.

– In the “Mode” category, click  to create a translation defined by an increment vector.

– In the “Vector” category, click . Specify the direction and the distance by snapping to the **A1** column base, and then the **A1** column top.

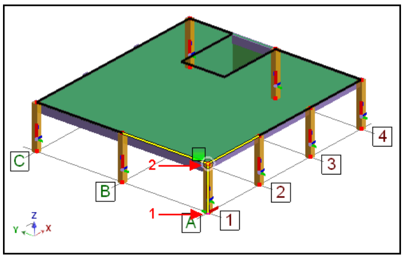


Figure 46: Specifying the translation vector

5. Click **Advanced** for more options. The “Option” panel appears.

6. Select the **Destination system** option and from the drop-down list, select the **Columns** subsystem of **FirstLevel**.

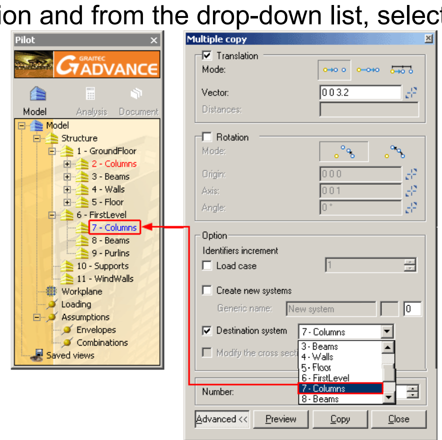


Figure 47: Selection of the destination subsystem

Verify the number displayed in the Pilot to be sure it is the right subsystem.

7. Click **Preview** to see how the settings are applied.

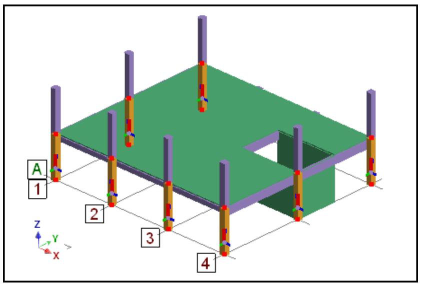
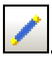


Figure 48: The columns of **FirstLevel**

8. Click **Copy** and close the dialog box.

The copied elements are created in the specified subsystem - the **Columns** subsystem of **FirstLevel**.

Creating beams on FirstLevel

1. In the Pilot, select the **Beams** subsystem of **FirstLevel**.
2. On the **Modeling** toolbar, on the **Create a linear element** flyout, click .
3. In the Properties window, make the following settings:

   <u>Material</u>

   – Select the **C 16/20** material code.

   <u>Cross section</u>

   – From the "Extremity 1" drop-down list select the **R20*35** cross section.
4. Create a linear element between the tops of the **A1** and **A4** columns.

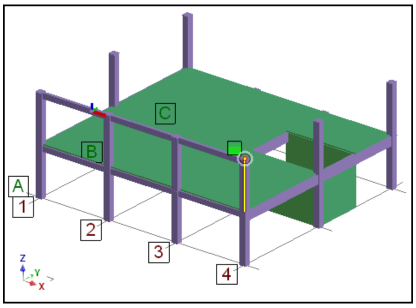


Figure 49: Creating a beam in the first level

5. Press <**Esc**> to quit the drawing function.

For a clear view of the **FirstLevel** system, hide the other systems of the structure: in the Pilot, right click **FirstLevel** and select **Isolate** from the context menu.

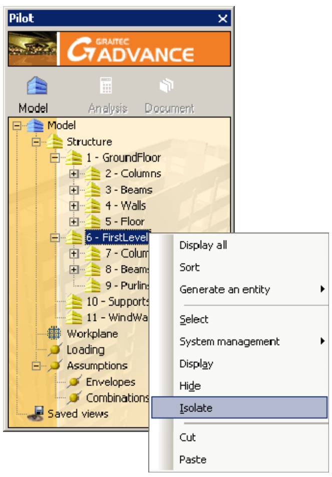


Figure 50: Isolating the **FirstLevel** system

Only the current system is displayed:

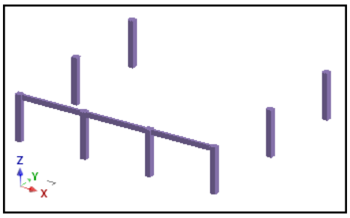


Figure 51: The current system is isolated

Subdividing a linear element

1. In the work area, select the beam created in **FirstLevel**.
2. On the **CAD Modifications** toolbar, click .
3. Enter **2** on the command line, to split the element in two equal elements.

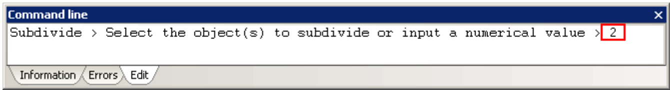


Figure 52: “Command line” - Subdividing a linear element

4. Press <**ENTER**>. The beam is split.

Rotating the beams

Next, rotate the beams by 9 degrees.

1. Click a beam to select it (Figure 53).

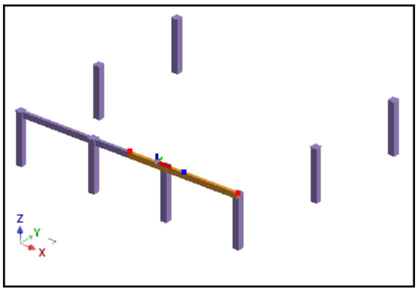
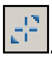


Figure 53: Selection of the first beam

2. On the **CAD Modifications** toolbar, click .
3. In the “Move” dialog box select **Rotation** and unselect **Translation**.
4. Set the rotation parameters:
   – In the “Origin” category, click . Specify the rotation origin by snapping to the end of the beam (Figure 54).

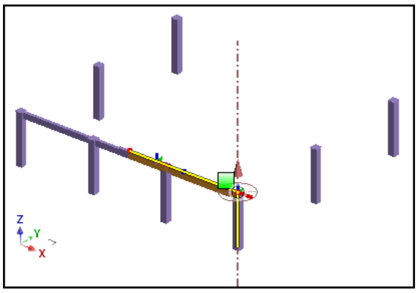


Figure 54: Specifying the rotation origin

– In the “Axis” category, click . Click the top of the **C4** column to specify the rotation axis.

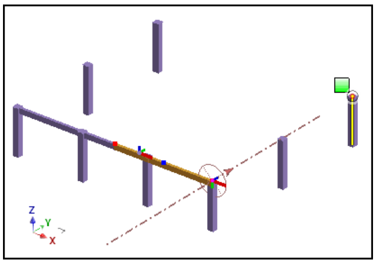


Figure 55: Specifying the rotation axis

– In the “Angle” category, enter **9**.

5. Click **Preview** to see how the settings are applied.

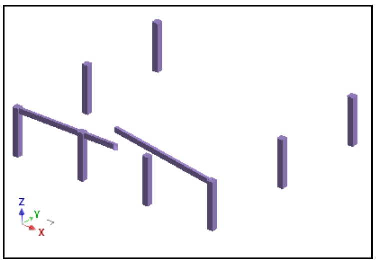


Figure 56: The rotated beam

6. Click **Move** and close the dialog box.

7. Press <**Esc**> to unselect the beam.

Using the same process, rotate the other beam by -9 degrees:

1. Select the other beam (Figure 57).

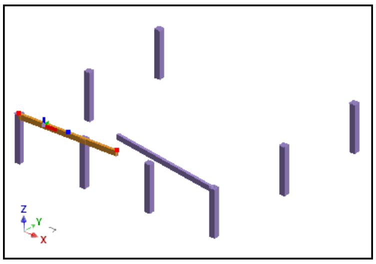


Figure 57: Selection of the second beam

2. On the **CAD Modifications** toolbar, click .

3. In the “Move” dialog box select **Rotation**.

4. Set the rotation parameters:

– In the “Origin” category, click . Specify the rotation origin by snapping to the end of the beam, as shown in the following figure:

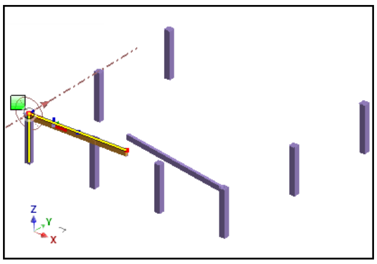


Figure 58: Specifying the rotation origin

The rotation axis is currently defined on the (**0, 1, 0**) coordinates.

– Enter **-9** in the “Angle” category to set the rotation angle.

5. Click **Preview** to see how the settings are applied.

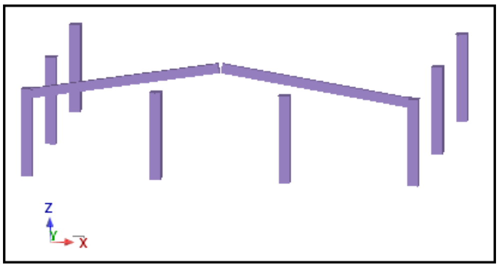


Figure 59: The result of the rotation of the two beams

6. Click **Move** and close the dialog box.

7. Press <**Esc**> to unselect the beam.

Adjusting element lengths

Press <**ALT** + **1**> for a front view.

Notice the gap between the ends of the beams.

---

*To zoom in on a certain area of the model, define a zoom window: press <**ALT + W**> (notice the cursor shape changes), click on the drawing area to define the upper left corner of the zoom window and drag the cursor to the right. Click to define the lower right corner of the zoom window.*

Adjust the beams length so that they link to each other:

1. Click the two beams to select them (Figure 60).

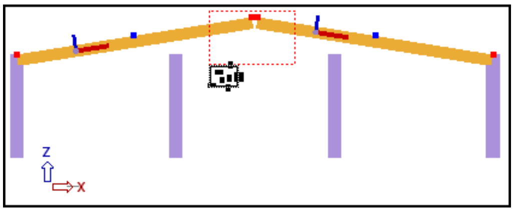


Figure 60: Selection of the beams

2. On the **CAD Modifications** toolbar, click .

3. Click each beam. The beams are extended and linked.

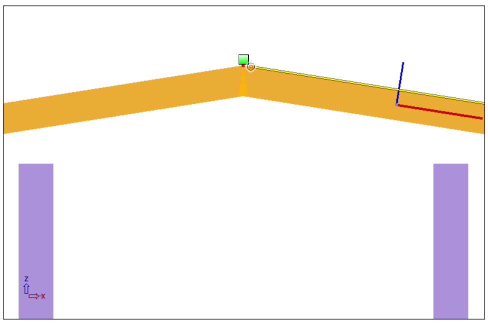
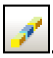


Figure 61: Adjusting the beams length

<u>Extend the columns so that they link to the beams</u>:

Make sure the two beams remain selected. This way they are a reference for the next adjustment.

1. On the **CAD Modifications** toolbar, click .

Double click the scroll mouse button to fit the view of the model on the screen.

2. Click the two columns.

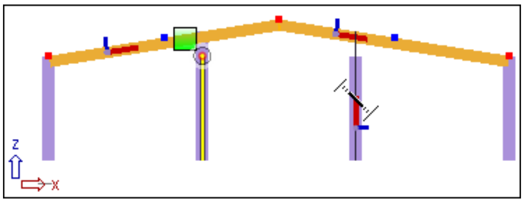


Figure 62: Extending the columns

3. Press <**Esc**> to disable the "Trim or extend" tool.

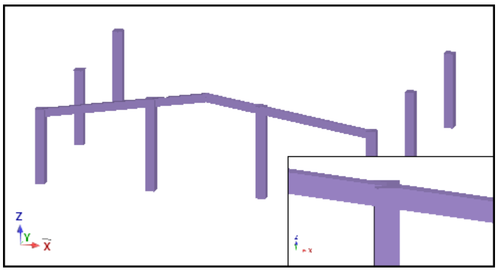


Figure 63: The two columns linked to the beams

Creating structure elements by multiple copy

Next, create the rest of the first level elements.

Press <**ALT + R**> and rotate for a 3D view.

1. Select the two columns and the two beams (Figure 64).

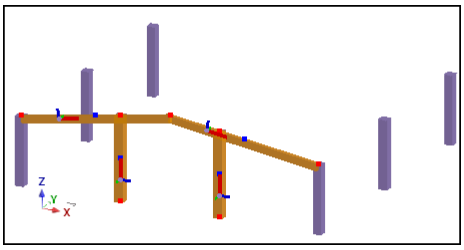


Figure 64: Selection of the beams and the columns

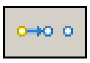
2. On the **CAD Modifications** toolbar, click .

3. In the "Multiple copy" dialog box, select the **Translation** option.

4. Set the translation parameters:

– In the "Option" panel, unselect the **Destination system** option and click **Advanced** to hide the panel.

– In the "Number" field, input **2** to generate two copies.

– In the "Mode" category, click  to create a translation defined by an increment vector.

– In the “Vector” category, click . Specify the direction and the distance by snapping to the tops of the **A4** and **B4** columns of **FirstLevel**.

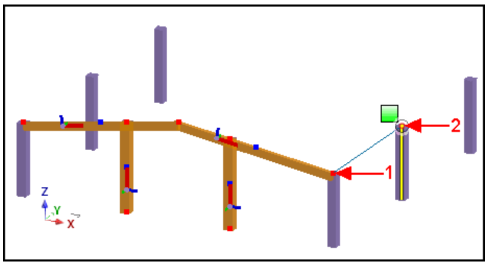


Figure 65: Specifying the copy coordinates

5. Click **Preview** to see how the settings are applied.

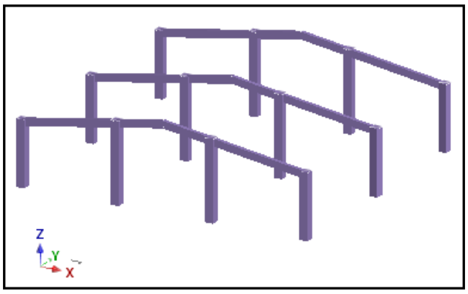


Figure 66: The result of the copy

6. Click **Copy** and close the dialog box.

The columns are created in the **Columns** subsystem and the beams in the **Beams** subsystem of **FirstLevel**. (Each copied element is created in the system to which the original element belongs.)

On the **Filters and selection** toolbar, click to display all the elements of the structure.

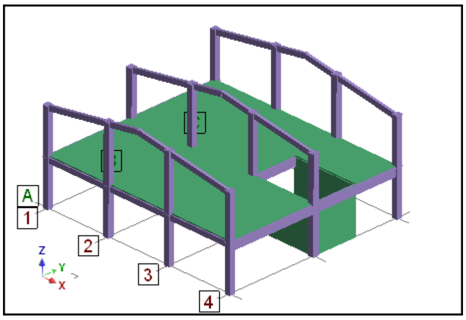


Figure 67: Displaying the whole structure

Creating purlins

1. In the Pilot, select the **Purlins** subsystem.

2. On the **Modeling** toolbar, on the **Create a linear element** flyout, click .

3. In the Properties window, make the following settings:

<u>Material</u>

– Select the **S235** material code.

<u>Cross section</u>

– From the “Extremity 1” drop-down list select the **UPE160** cross section.

4. Create two purlins:

– The first one by snapping to the top of the **A4** and **B4** columns of the first floor:

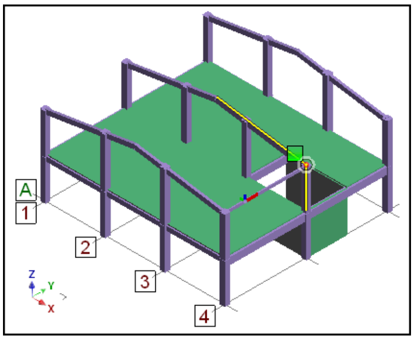


Figure 68: Creating the first purlin

– The second one by snapping to the top of the **B4** and **C4** columns.

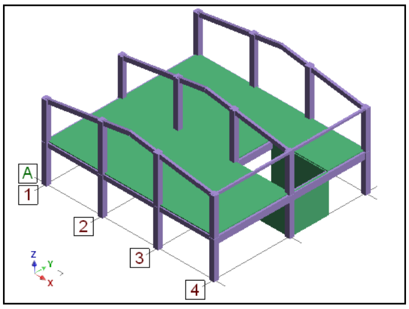


Figure 69: The two purlins

5. Press <**Esc**> to quit the drawing function.

Setting the purlin properties

1. Select the two purlins.

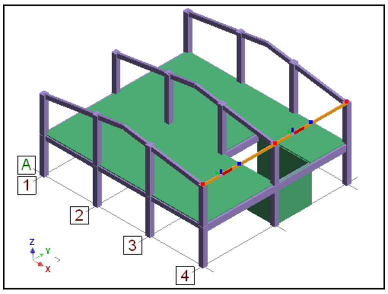


Figure 70: Selection of the purlins

2. In the Properties window, make the following settings:

Cross section

– Expand "Eccentricity", click **Option** and select **(0, z+)** from the drop-down list, to define the position of the cross section relative to its own axis.

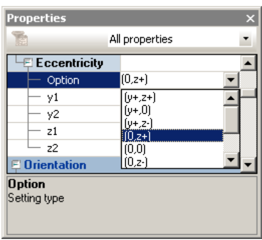


Figure 71: Setting the **Eccentricity**

Orientation

– In the “Orientation” category, enter **9** in the “Angle” field.

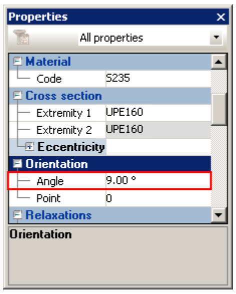


Figure 72: Setting the purlins orientation angle

Releases

– Expand the “Total releases” category. To enable the rotation around the **Y**-axis on both extremities, expand **Extremity 1** and **Extremity 2** and enable the **Ry** option for both extremities. The rotation of the object's extremities on the **Y**-axis is unlocked.

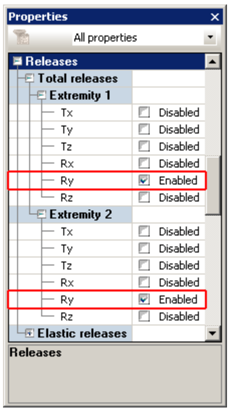


Figure 73: Enabling relaxations for rotating on **Y**-axis

Isolate the first level for a better view: in the Pilot, right click **FirstLevel** and select **Isolate**.

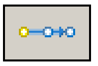
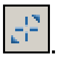
Creating purlins using copy by translation defined by a total vector

Next, create 5 copies of each purlin on the length of a beam.

1. Select the two purlins.

2. On the **CAD Modifications** toolbar, click .

3. In the “Multiple copy” dialog box select the **Translation** option.

4. Set the translation parameters:
   - In the “Number” field, input **5** to generate five copies.
   - In the “Mode” category, click  to create a translation defined by a total vector.
   - In the “Vector” category, click . Specify the direction and the distance by snapping to the endpoints of a beam (Figure 74).

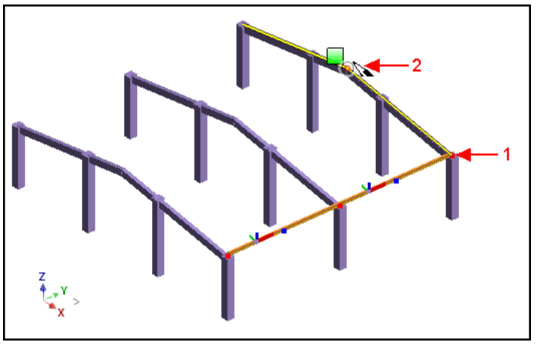


Figure 74: Specifying the copy coordinates

5. Click **Preview** to see how the settings are applied.

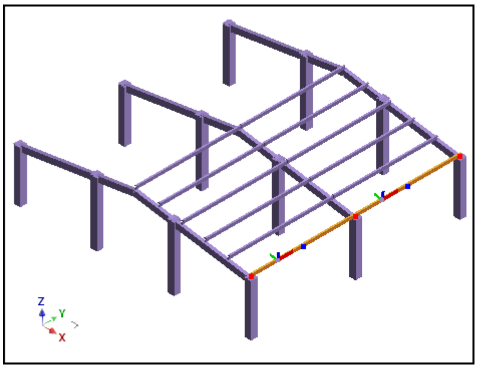


Figure 75: Purlins

6. Click **Copy** and close the dialog box.
7. Press <**Esc**> to unselect the purlins.

Press <**ALT + 1**> for a front view.

Zoom the image: press <**ALT + W**>, notice that the cursor shape changes. Define the zoom window from left to right to zoom in.

Notice the position of the purlins and the orientation of their cross sections.

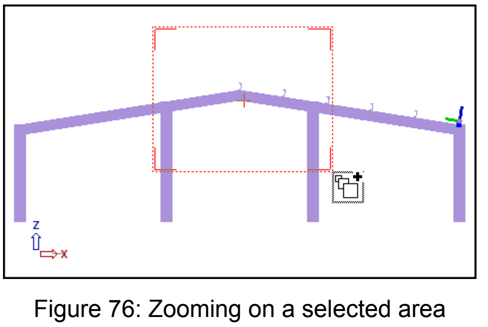


Figure 76: Zooming on a selected area

Moving the purlins a user-defined distance

Press <**ALT + 5**> for a 3D view.

1. Select the first two purlins.

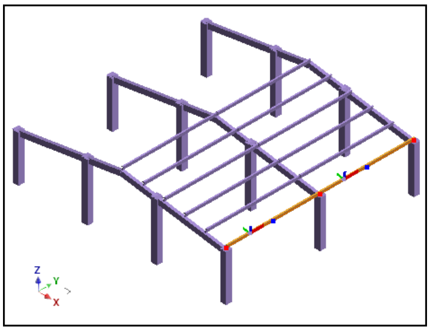
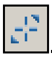


Figure 77: Selection of the purlins

2. On the **CAD Modifications** toolbar, click .

3. In the “Move” dialog box select **Translation** and unselect **Rotation**.

4. Set the direction of the translation:

- In the “Vector” category, click .
- Snap to the end of a beam (notice that a green square appears) and click (Figure 78).

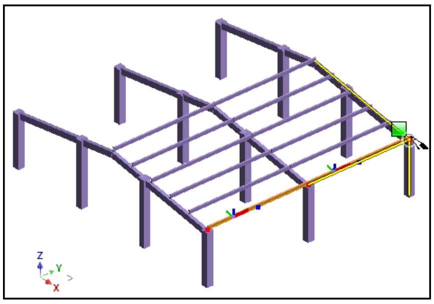


Figure 78: Specifying the first coordinate

- Move the cursor in the direction of the other end of the beam and snap the cursor without clicking.

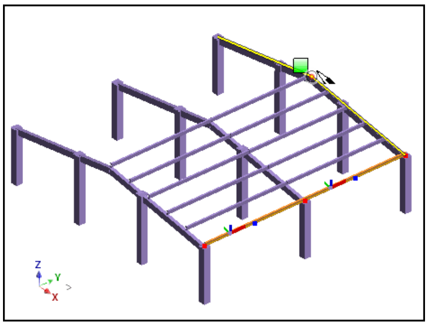


Figure 79: Specifying the direction

5. On the command line, enter **0.2** to move the purlins 0.2 m in the specified direction.

6. Press <**ENTER**>.

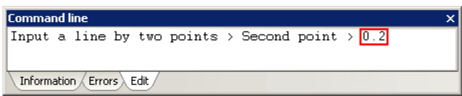


Figure 80: Specifying the distance on the “Command line”

7. Click **Preview** to see how the settings are applied.

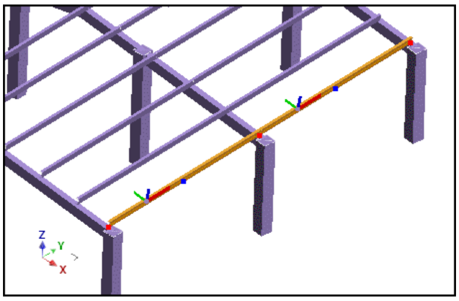


Figure 81: The purlins moved 0.2 m in the specified direction

8. Click **Move** and close the dialog box.
9. Press <**Esc**> to unselect the purlins.

Using the same process, move the last purlins:

1. Select the last two purlins.

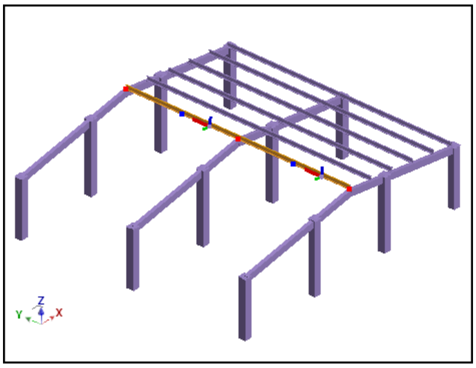
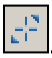


Figure 82: Selection of the last two purlins

2. On the **CAD Modifications** toolbar, click .
3. In the “Move” dialog box select the **Translation** option.
4. Set the direction of the translation:
   - In the “Vector” category, click .
   - Snap to the top of a beam and click (Figure 83).

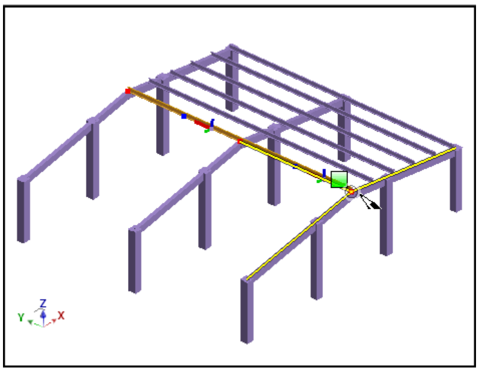


Figure 83: Specifying the first coordinate

– Move the cursor in the direction of the other end of the beam and snap the cursor without clicking.

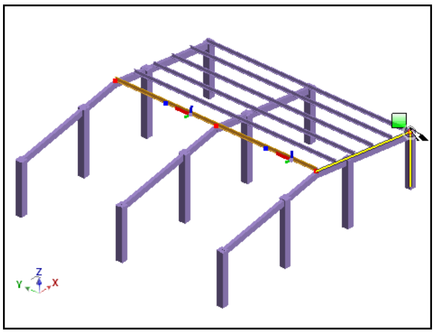


Figure 84: Specifying the direction of the vector

5. On the command line, enter **0.2**.

6. Press <**ENTER**>.

7. Click **Preview** to see how the settings are applied.

8. Click **Move** and close the dialog box.

9. Press <**Esc**> to unselect the purlins.

Copying the purlins by symmetry

Press <**ALT + 3**> for a top view.

1. Select all the purlins.

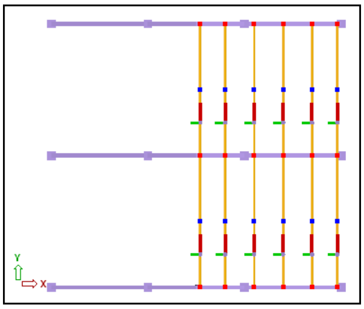
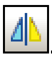


Figure 85: The top view makes the selection of the purlins easier

Press <**ALT + 5**> for a 3D view.

2. On the **CAD Modifications** toolbar, click .

3. In the “Symmetries” dialog box select **Copy**.

4. Set the copy parameters:

– From the “Plane” drop-down list, select **YZ**.

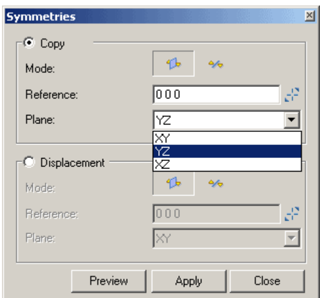


Figure 86: Selection of the plane of the copy

– In the "Reference" category, click . Specify the reference point as the intersection of the beams (Figure 87).

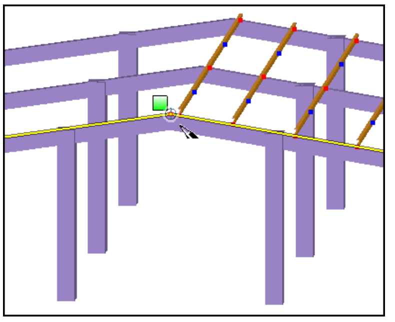


Figure 87: Specifying the reference point

5. Click **Preview** to see how the settings are applied.
6. Click **Apply** and close the dialog box.

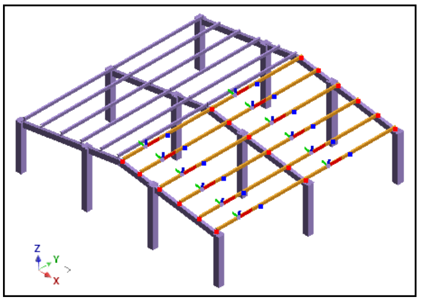


Figure 88: The purlins created by symmetry copy

Press <**Esc**> to unselect all elements and zoom to view the orientation of the cross sections. Notice that all purlins have the same orientation.

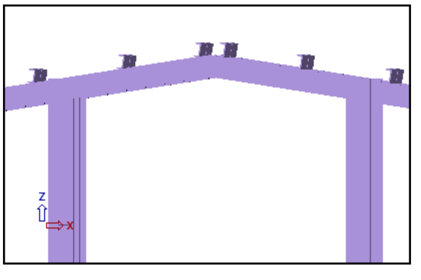


Figure 89: The orientation of the cross sections

Changing the orientation of the purlins cross sections

1. Press <**ALT + 3**> for a top view and select the newly created purlins.

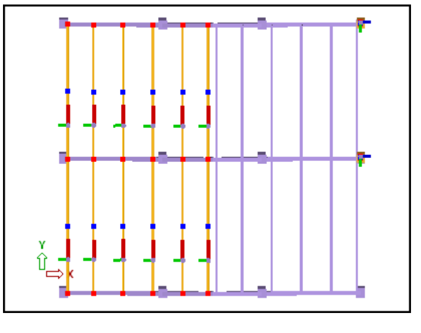


Figure 90: Selection of the newly created purlins

2. In the Properties window, make the following settings:

   Orientation

   – Enter **171** in the “Angle” field.

   Cross section

   – Expand “Eccentricity”, click **Option** and select (**0, z-**) from the drop-down list.

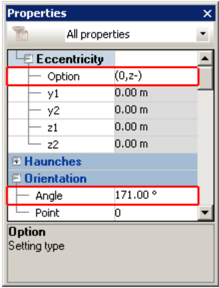


Figure 91: Setting the eccentricity and the orientation angle of the purlins

Press <**ALT + 1**> for a front view and zoom. Notice that the orientation of the cross sections changed.

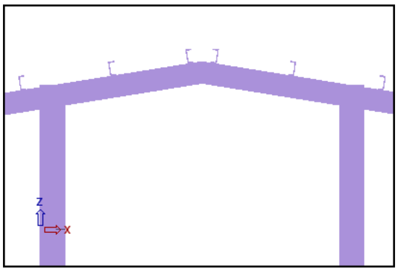
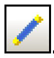


Figure 92: The new orientation of the cross sections

Press <**ALT + 5**> for a 3D view and click  to display all the elements of the model.

Creating transverse beams on the first level

1. In the Pilot, select the **Beams** subsystem of **FirstLevel**.

2. On the **Modeling** toolbar, on the **Create a linear element** flyout, click .

3. In the Properties window, make the following settings:

   Material

   – Select **C16/20** material code.

   Cross section

   – From the “Extremity 1” drop-down list, select **R20*30** cross section.

4. Create a beam along the **4** axis connecting the top extremities of the **A4** and **B4** columns of the first level.

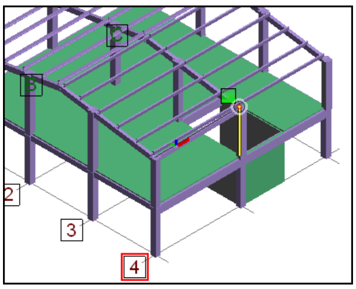


Figure 93: Creating the first beam on the **4** axis

5. Create another beam between the tops of the **B4** and **C4** columns.

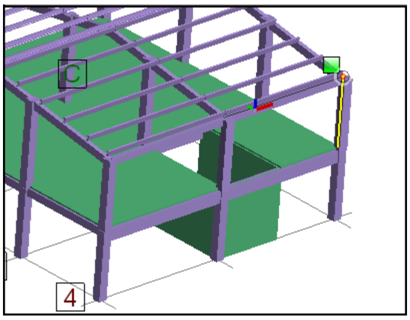


Figure 94: Creating the second beam on the **4** axis

6. Using the same process, create six beams along the other axes (**1**, **2** and **3**), connecting the top extremities of the columns.

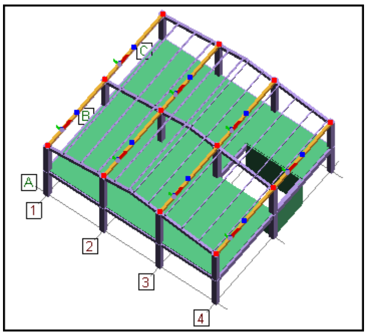


Figure 95: The beams of the first level of the structure

7. Press <**Esc**> to quit the drawing function.

Creating fixed point supports

On the **Predefined views** toolbar, click  for a perspective view.

Press <**ALT + R**> and rotate to get a view of the base of the structure.

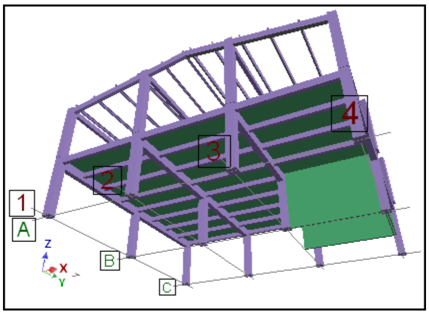


Figure 96: The base of the structure

1. In the Pilot, select the **Supports** system.

2. On the **Modeling** toolbar, on the **Create a point support** flyout, click  to create a rigid point support.

3. In the Properties window, on the “Restraints” category, from the “Type” drop-down list, select **Fixed**.

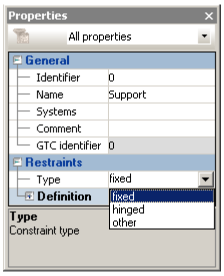


Figure 97: Selection of the restraints type

4. Place a support at the base of each column.

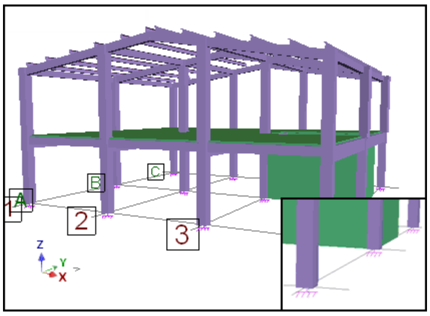


Figure 98: Rigid supports at the base of the columns

5. Press <**Esc**> to quit the drawing function.

Creating fixed linear supports

1. On the **Modeling** toolbar, on the **Create a linear support** flyout, click  to create a rigid linear support.
2. In the Properties window, on the “Restraints” category, from the “Type” drop-down list, select **Fixed**.
3. Use the scroll mouse button to zoom on the base of the walls and create three linear supports by snapping to the base endpoints of each wall.

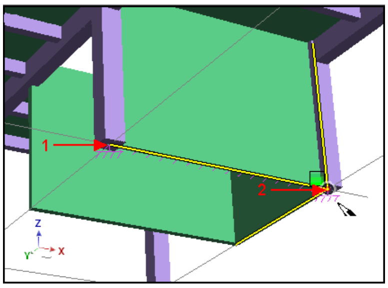


Figure 99: Creating a linear support at the base of a wall

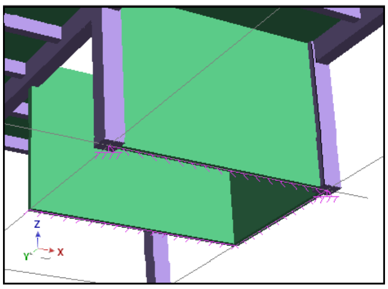


Figure 100: Linear supports at the base of the walls of the structure

4. Press <**Esc**> to quit the drawing function.

Step 2: Use display settings for easier handling of the model

In this step, set the elements display color by certain categories such as material and cross section.

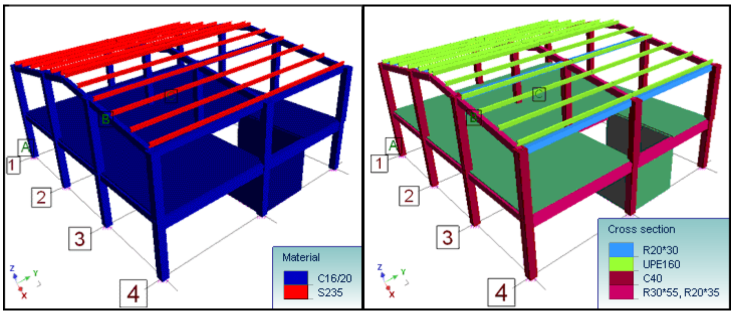


Figure 101: The structure's elements highlighted depending on their material and cross section

Displaying structure elements by material

1. From the main menu, select **Display > Display settings**. The "Display settings" dialog box appears.
2. In the "Colors" area, from the "Display by" drop-down list, select **Material**.
3. Enable the **Color legend** option to display the description of the colors used in the model.

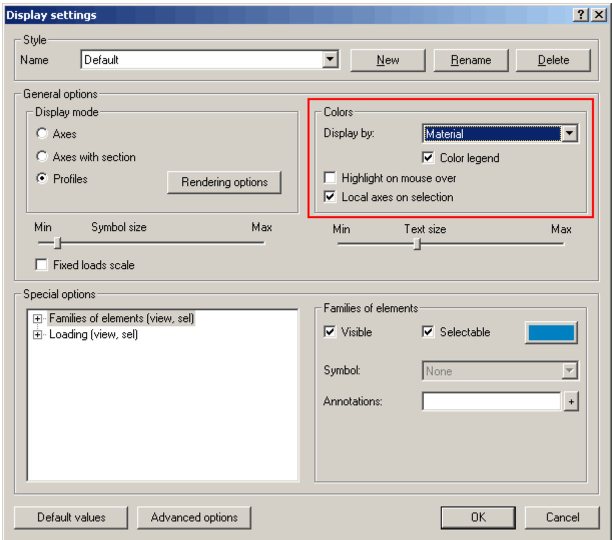


Figure 102: The "Colors" area from the "Display settings" dialog box

4. Click **OK**.

Press <**ALT + 5**> for a 3D view.

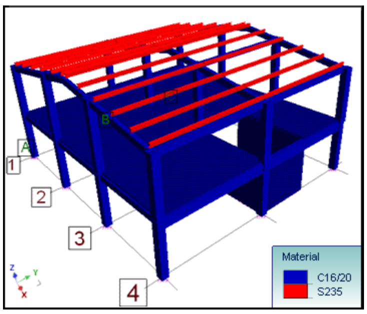


Figure 103: Each element is represented by a certain color, depending on the material

Displaying structure elements by cross sections

1. From the main menu, select **Display > Display settings**. The “Display settings” dialog box appears.

2. In the “Display settings” dialog box, from the “Display by” drop-down list select **Cross section**.

3. Click **OK**.

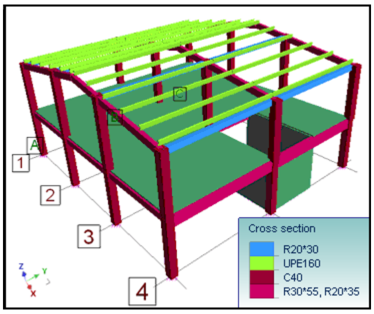


Figure 104: Each element is represented by a certain color, depending on its cross section

<u>Returning to the display by type of elements</u>:

1. Press <**ALT + X**> to open the “Display settings” dialog box.

2. In the “Colors” area, from the “Display by” drop-down list, select **Element family**. You can display/hide the “Legend” on the work area by selecting/unselecting the **Color legend** option.

3. Click **OK**.

Lesson 3: Creating windwalls and defining snow and wind supporting elements

In this lesson you will create windwalls to distribute loads on the supporting elements.

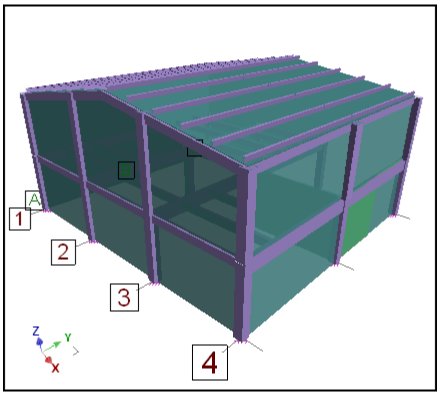


Figure 105: The windwalls of the structure

You will learn how to:

- Create windwalls.
- Define supporting elements.
- Define the load distribution towards the supporting elements.
- Use the rendering modes for an adequate view according to the process.

Before starting

Considering that in this step you will create windwalls that hide elements of the structure, it is recommended to modify the transparency level, so that all the elements of the structure can be viewed.

Modify the transparency of the structure:

1. Press <**ALT + X**> to access the “Display settings” dialog box.
2. In the “General options” area click **Rendering options**.
3. In the “Rendering" dialog box, “Advanced parameters” area, drag the Transparency slider to the right for more transparency.

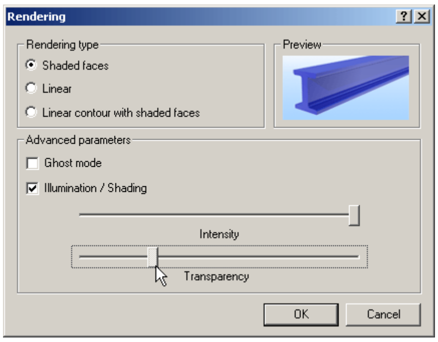


Figure 106: “Rendering” dialog box - Setting the transparency of the elements

4. Click **OK** to close the “Rendering” dialog box.
5. Click **OK** in the “Display settings” dialog box.

Step 1: Create windwalls

In this step, create windwalls on the profiles of the structure.

1. In the Pilot, select the **WindWalls** system.
2. On the **Modeling** toolbar, click  to create a windwall element.
3. Create a windwall by snapping to the endpoints of the first portal frame of the structure.

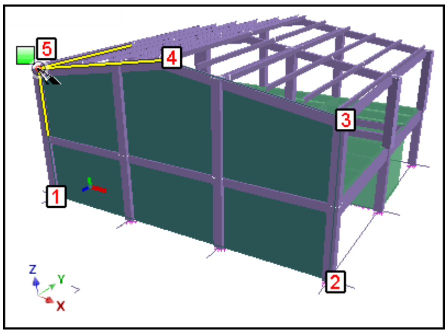


Figure 107: Creating the first windwall of the structure

4. Press <**ENTER**> to finish.

Using the same process, create windwalls on the other three sides of the structure.

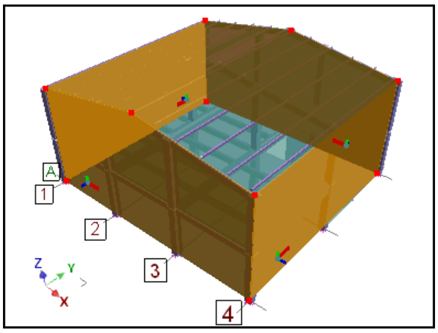


Figure 108: The windwalls on the sides of the structure

Step 2: Automatically generate windwalls

In this step, automatically generate windwalls on selected elements.

Press <**ALT + 5**> for a 3D view.

1. Select two opposite transverse beams of the first level, as shown in Figure 109.
2. Right click in the work area and select **Windwalls / selection** from the context menu.

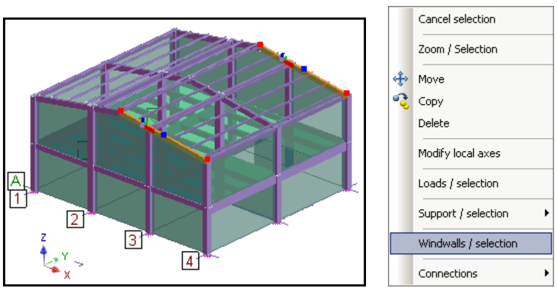


Figure 109: Automatically generating windwalls between two selected elements

3. Press <**Esc**> to unselect the beams.

Using the same process, generate a windwall between the other two beams.

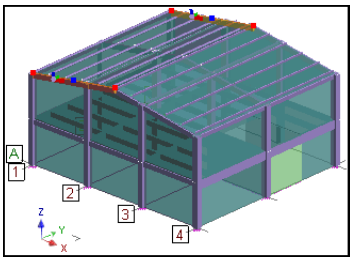


Figure 110: The windwalls of the structure

Step 3: Define supporting elements

In this step, define the supporting elements for wind and snow. Certain elements of the structure do not undertake wind and snow loads, therefore the “supporting element” option must be disabled.

Before starting, hide the windwalls: in the Pilot, right click the **WindWalls** system and select **Hide**.

Disable the beams of **FirstLevel** as supporting elements for snow and wind.

1. In the Pilot, click the **Beams** subsystem of **GroundFloor** and press <**SPACE**>. All objects of the subsystem are selected in the Pilot and in the work area.

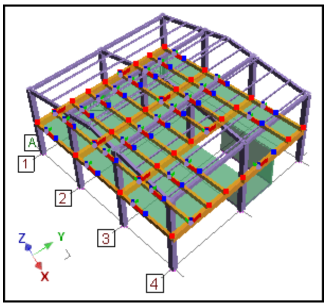


Figure 111: Selecting the elements of the **Beams** subsystem of **GroundFloor**

The command line displays the number and type of the selected objects.

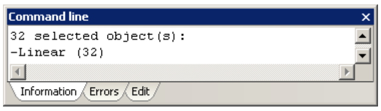


Figure 112: “Command line” - the objects from the selected system

2. The Properties window displays the common properties of the selected objects. Go to the “Snow and Wind” category and disable the **Supporting element** option. The selected elements are no longer supporting elements for snow and wind.

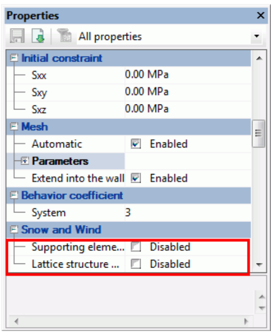


Figure 113: “Snow and Wind” category, in the Properties window

Using the same process, disable the **Walls** subsystem of **GroundFloor** and the **Beams** subsystem of **FirstLevel** as supporting elements for snow and wind.

Step 4: Display the direction of the load distribution

In this step you will use the rendering options to view the direction of the load distribution on the windwalls.

1. In the Pilot, right click the **WindWalls** system and select **Isolate** from the context menu to display only the windwalls.

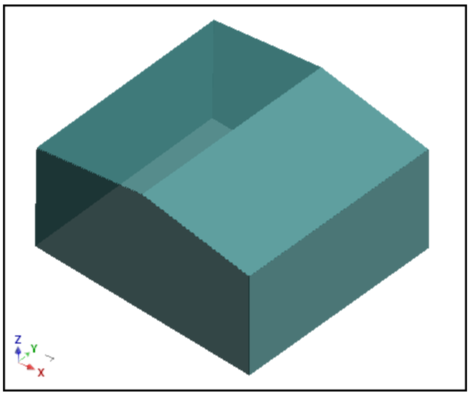


Figure 114: Isolated **WindWalls** system

2. On the **Rendering** toolbar, click  to enable the “Axes with sections” rendering mode that allows the display of the direction of the load distribution on the windwalls.

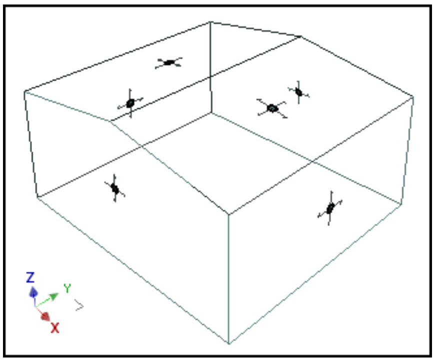


Figure 115: The windwalls direction of the load distribution viewed in “Axes with sections” rendering mode

Step 5: Set the windwalls load distribution direction

In this step, set the direction of the loads on each windwall of the structure.

1. Select one of the top windwalls by clicking on the load distribution symbol (Figure 116).

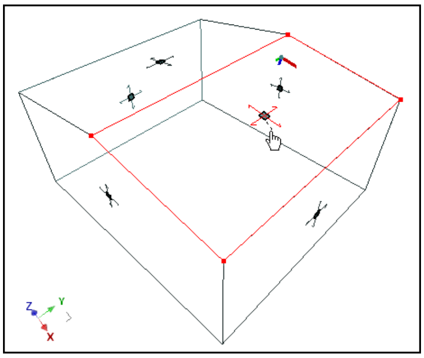


Figure 116: Selecting a windwall in the “Axes with section” rendering mode

2. In the Properties window, in the "Load distributions" category, set the "Span direction" to **x** to define the direction of the load on the **x**-axis.

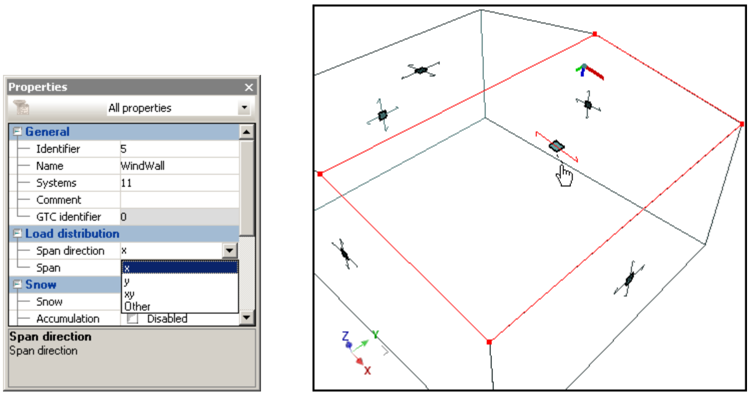


Figure 117: Defining the direction of the load distribution on the **x**-axis

3. Press <**Esc**> to unselect the windwall.

Using the same process, select the span direction on the **x**-axis for all windwalls.

<u>Return to the profiles rendering mode</u>:

1. On the **Filters and selection** toolbar, click to display all the elements.

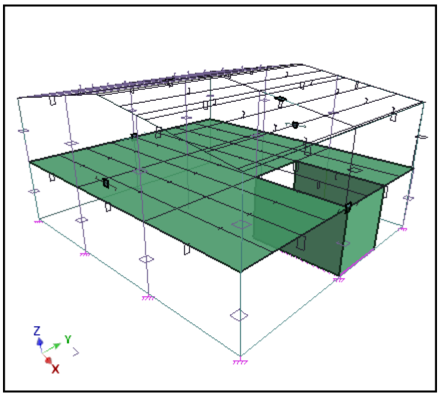


Figure 118: The whole structure in the "Axes with sections" rendering mode

2. On the **Rendering** toolbar, click to display the profiles of the elements.

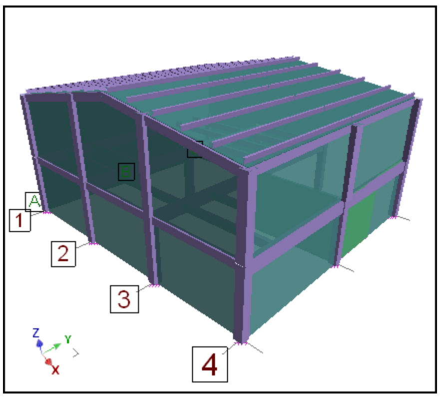


Figure 119: The whole structure in the "Profiles" rendering mode

Lesson 4: Defining Self Weight Loads and Live Loads

In this lesson you will generate loads on the structure's elements. The loads are grouped in load cases, which correspond to certain case families. A case family groups a set of load cases of the same nature.

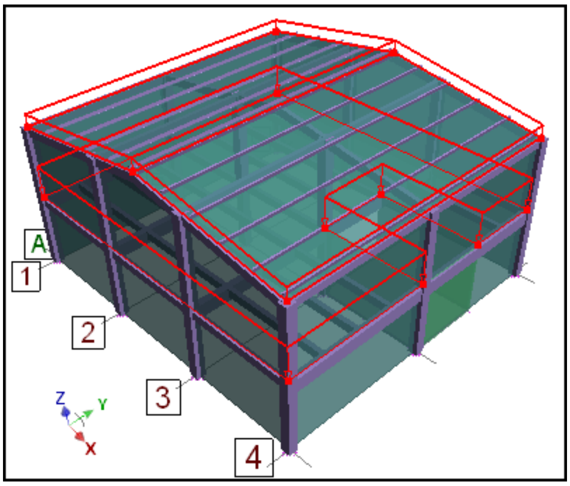


Figure 120: The dead loads and live loads of the structure

You will learn how to:

- Generate the self weight loads on the entire structure.
- Generate live loads on selected elements.

Step 1: Generate the self weight loads on the entire structure

In this step, you will create case families of dead loads and generate the self weight load of the structure.

Creating the self weight loads

1. In the Pilot, right click **Loading** and select **Create a case family** from the context menu.

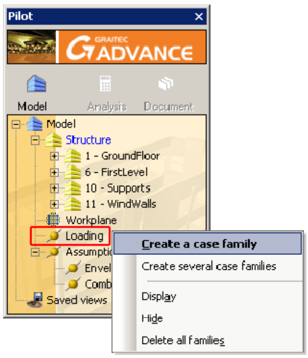


Figure 121: Creating a loads case family

2. In the "Create a load case family [Solicitation]" dialog box, select **Dead Loads** and click **OK**.

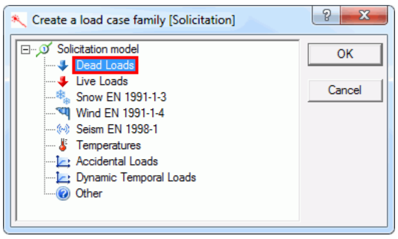


Figure 122: "Create a load case family" dialog box - **Dead Loads** case family

A dead loads case family and a dead load case (**G**) are created in the Pilot, in **Loading**.

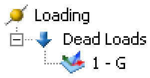


Figure 123: The dead loads case family and the dead load case

Renaming a load case

1. In the Pilot, expand **Dead Loads**, right click the **G** load case and select **Rename** from the context menu.
2. Type **SelfWeight** and press <**ENTER**>.

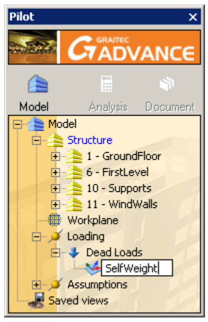


Figure 124: Renaming the load case

In the Properties window, notice the “Gravity field” category indicating the **X**, **Y** and **Z** values of self weight intensity.

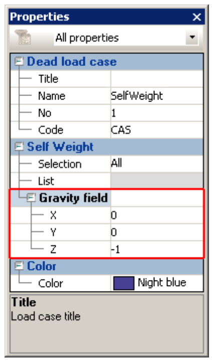


Figure 125: The “Gravity field” category, in the Properties window

***Note**: The **X**, **Y** and **Z** values of self weight intensity are given by default at the creation of the self weight load case.*

Generating a self weight load on the top windwalls

1. Select the top windwalls.
2. Right click in the work area and select **Load / selection** from the context menu.

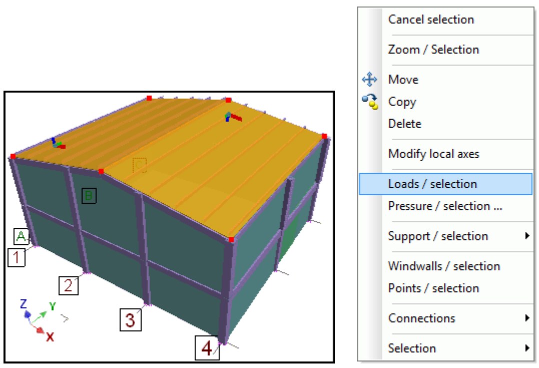


Figure 126: Top windwalls

3. In the Properties window, in the “Definition” category set the load component along the **Z**-axis (**FZ**) to **-0.2 kN**.

4. Click **OK**.

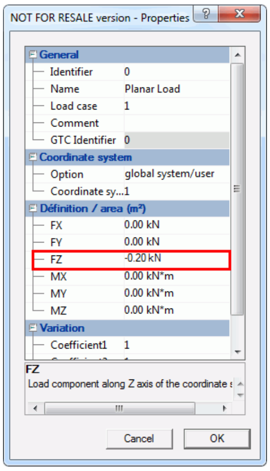


Figure 127: Entering the **FZ** value for the self weight load case

Displaying the loads

In the Pilot, expand the **SelfWeight** load case and click one of the planar loads created for each of the two windwalls. The graphical representation of the selected load is highlighted in the work area.

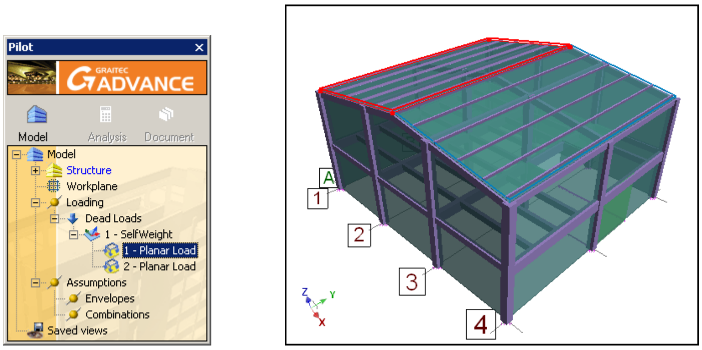


Figure 128: The graphical representation of the dead load

Press <**Esc**> to unselect all elements.

For a better representation of the load, adjusted to the model scale: right click in the work area and select **Loads auto-scale** from the context menu.

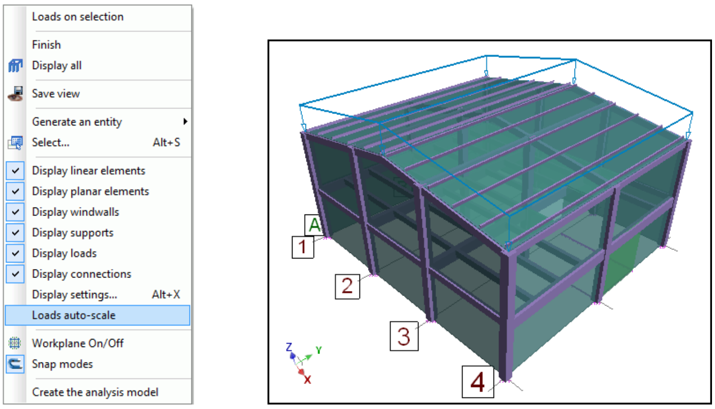


Figure 129: The planar loads adjusted to the model scale

Note: *The **Loads auto-scale** command updates the display of the loads according to the structure's size and relative to the other loads intensities.*

Step 2: Generate live loads on selected elements

In this step, you will create live loads case families and generate a live load on the floor of the first level of the structure.

Generating a live load on the floor

1. In the Pilot, right click **Loading** and select **Create a case family** from the context menu.
2. In the "Create a load case family [Solicitation]" dialog box, select **Live Loads**.

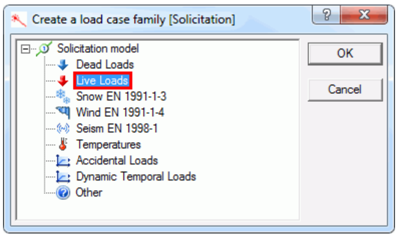


Figure 130: Selection of the **Live Loads** case family from the "Create a load case family" dialog box

3. Click **OK**.

A live loads case family is created in the Pilot, in **Loading**, and a live load case (**Q**) is created by default.

4. In the Pilot, right click the live load case (**Q**) and select **Rename** from the context menu. Rename it **LiveLoads**.

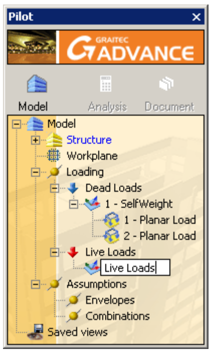


Figure 131: Renaming the live load

5. In the Pilot, right click the **GroundFloor** system and select **Isolate**. Double click the **Dead Loads** to hide them.
6. Select the floor, then right click the work area and select **Load / selection** from the context menu.

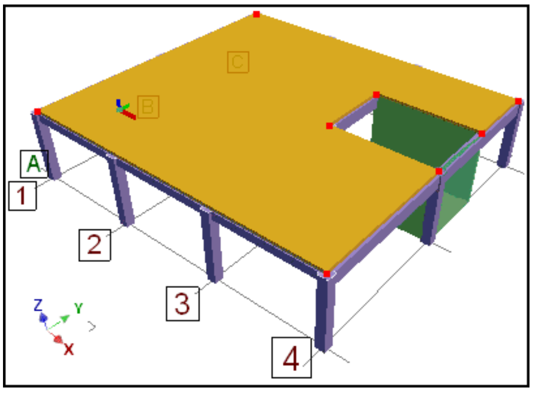


Figure 132: Selecting the floor

7. In the Properties window, in the “Definition” category, set the load component along the **Z**-axis (**FZ**) to **-2 kN**.

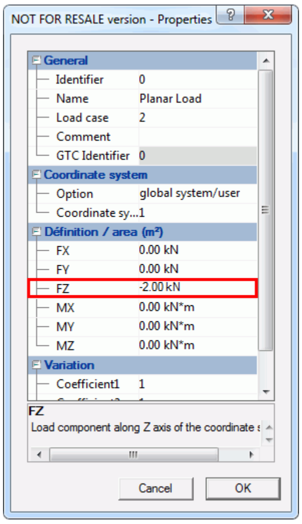


Figure 133: Entering the **FZ** value

8. Click **OK** to apply.
9. Pres <**Esc**> to unselect the floor.

Right click in the work area and select **Loads auto-scale**.

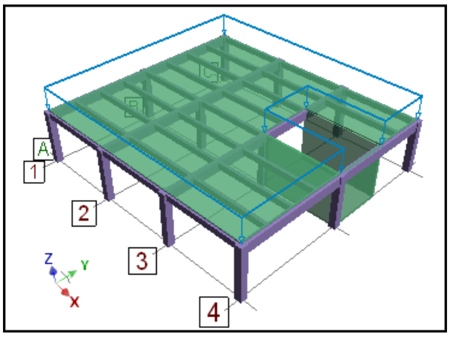


Figure 134: The live load on the floor

Generating a live load on the top of the structure

On the **Filters and selection** toolbar, click  to display all the elements of the structure.

1. Select the top windwalls.

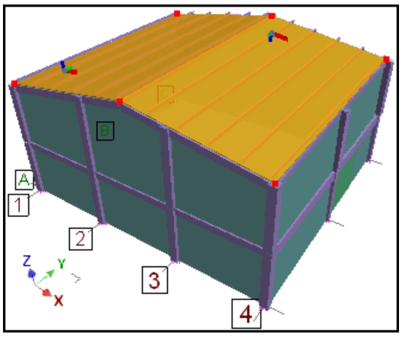


Figure 135: Selecting the top windwalls of the structure

2. Right click in the work area and select **Load / selection** from the context menu.

***Note**: On the **Modeling** toolbar, a drop-down list displays the current load case.*

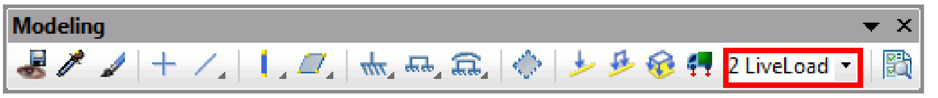


Figure 136: The current load case displayed in the **Modeling** toolbar

3. In the Properties window, in the “Definition” category, enter **-0.75 kN** for the value of the load component along the **Z**-axis (**FZ**).
4. Click **OK** to apply.
5. Press <**Esc**> to unselect the windwalls.

Right click in the work area and select **Loads auto-scale**.

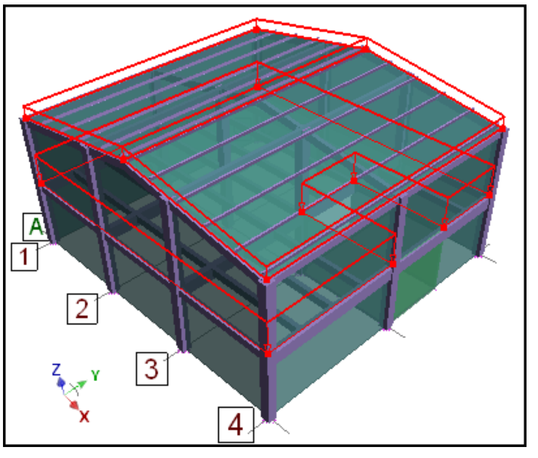


Figure 137: The dead loads and live loads of the structure

Lesson 5: Automatic creation of wind loads

In this lesson you will create wind load cases.

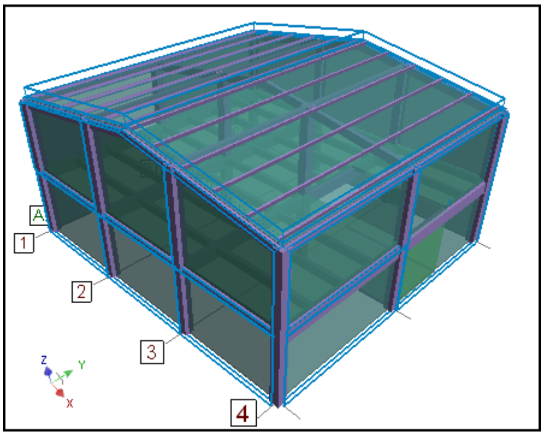


Figure 138: Wind loads representation on the structure

You will learn how to:

- Create wind load cases.
- Define the building's site, height and the wind pressure.
- Automatically generate wind loads.
- Manage the display of loads grouped in load cases.

Step 1: Create a wind load case family

Next, you will create a wind load case family and a wind load case.

1. In the Pilot, right click **Loading** and select **Create a case family** from the context menu.
2. In the “Create a case family [Solicitation]” dialog box, select **Wind EN 1991-1-4** and click **OK**.

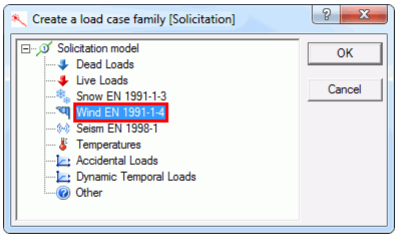


Figure 139: Selecting the wind case family

A wind case family is created in the Pilot, in **Loading**; also, a wind load case (**V**) is created by default.

Step 2: Set the properties of the wind case family

Next, you will set the wind case family properties according to the structure location.

1. In the Pilot, select the **Wind EN 1991-1-4** case family.

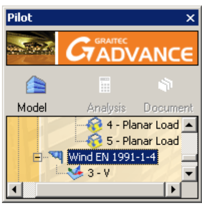


Figure 140: Selecting the **Wind EN 1991-1-4** case family

2. In the Properties window, set the properties of the **Wind EN 1991-1-4** case family:

   Dynamic base pressure

   – Enter a **24m/s** wind speed.

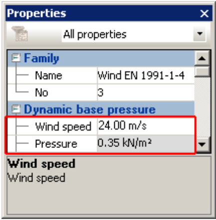


Figure 141: "Dynamic base pressure" values

Measure the height of the structure

From the main menu, select **Edit > Coordinates > Length**. The measuring tool appears. Click the highest point of the model and then on its projection on the ground.

***Note**: On the **Snap modes** toolbar, enable the **Perpendicular** snap mode to snap perpendicularly to an object.*

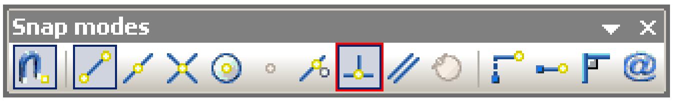


Figure 142: The enabled modes are highlighted in the **Snap modes** toolbar

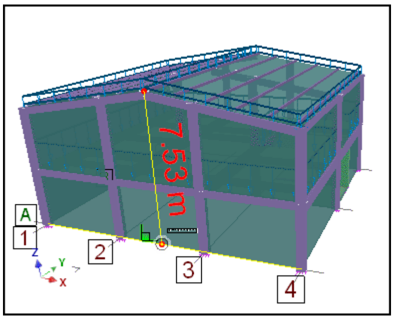


Figure: 143: Measuring the structure's height

*Press <**CTRL + D**> to access the measuring tool faster.*

The model has a height of **7.53 m**. Enter **8** (the approximate height of the structure) in the Properties window, in the "Dynamic base pressure" category, **Building height** field.

Step 3: Automatically generate wind loads

After making all the settings considering the location of the structure, you can automatically generate the wind loads.

In the Pilot, right click the **Wind EN 1991-1-4** case family and select **Automatic generation** from the context menu.

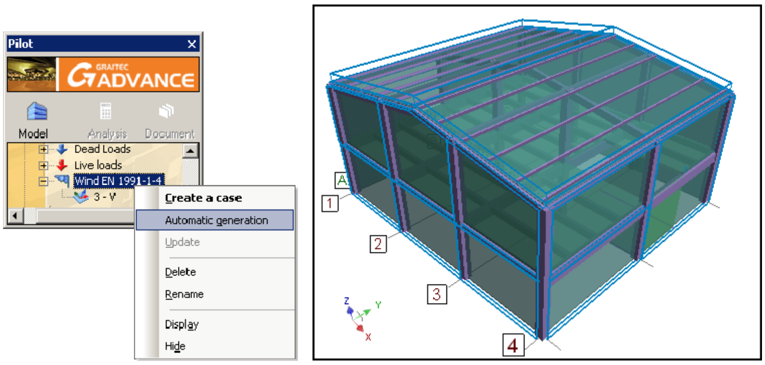


Figure 144: Automatic generation of the wind loads

The wind loads cases and wind loads are automatically generated.

Step 4: Display the wind loads from a load case family

For a clear view of the wind loads, display only the windwalls system: In the Pilot, right click the **WindWalls** system and select **Isolate** from the context menu.

Only the windwalls and the loads are displayed.

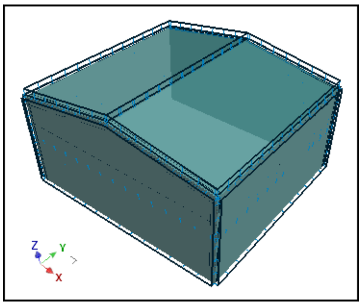


Figure 145: **WindWalls** system, isolated from the rest of the structure

To display only one load case, isolate it.

Displaying the Wind X+S load case

1. In the Pilot, right click the **Wind X+S** case and select **Isolate** from the context menu.

Right click in the work area and select **Loads auto-scale**. Rotate the image for a better view.

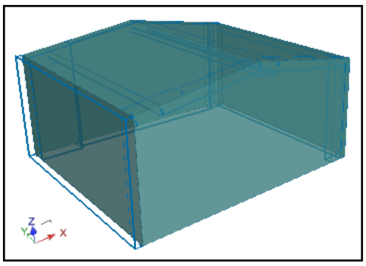


Figure 146: Isolating the **Wind X+S** load case

In the Pilot, double click any system, element of the structure, load case family or load to hide / display it in the work area. When hidden, the icon appears grey in the Pilot.

2. On the **Rendering** toolbar, click  to switch to “Axes with sections” rendering mode.

Press <**ALT + R**> to rotate the image. Notice the direction of the wind pressure.

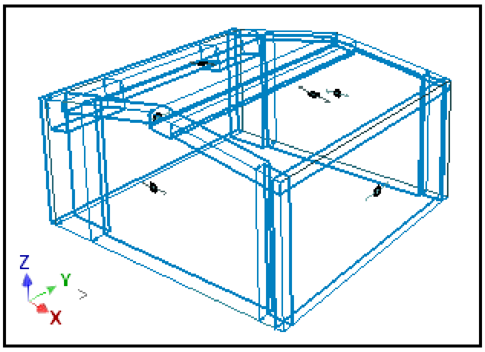


Figure 147: The **Wind +X** load case

Next, hide the **Wind X+S** load case and display **Wind X+D**.

Displaying the Wind X+D load case

In the Pilot, right click **Wind X+D** case and select **Isolate**.

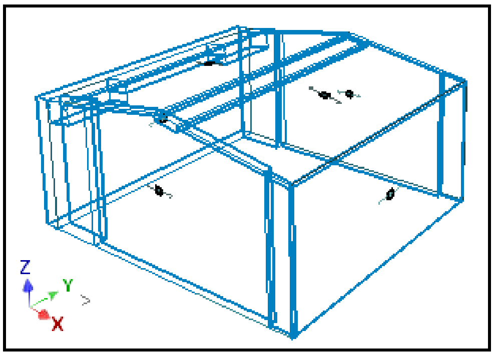


Figure 148: The **Wind -X** load case

<u>Return to the profiles rendering mode</u>:

1. On the **Rendering** toolbar, click  to display the profiles of the structure’s elements.

2. On the **Filters and selection** toolbar, click  to display all the elements and loads of the structure.

3. Right click in the work area and select **Loads auto-scale**.

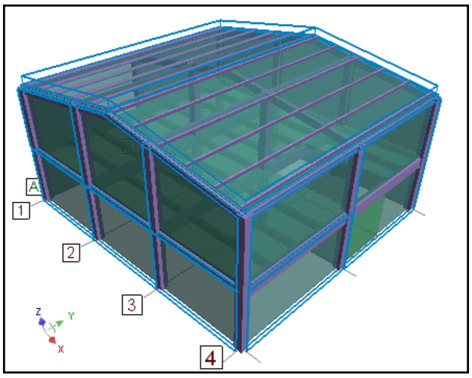


Figure 149: The wind loads of the structure in the “Profiles” rendering mode

Lesson 6: Automatic creation of snow loads

In this lesson you will automatically generate snow loads according to the current standards.

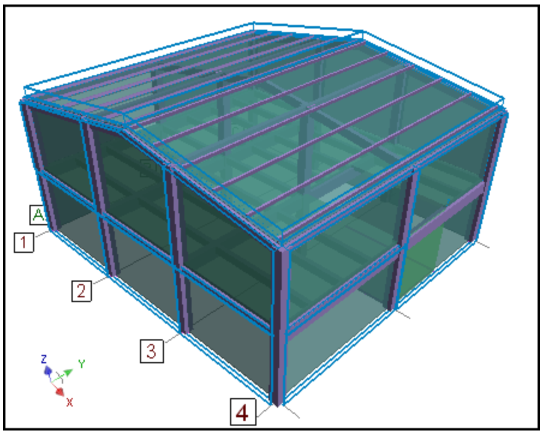


Figure 150: Automatically generated snow loads

You will learn how to:

- Create snow load cases.
- Define the snow pressure according to the building's location.
- Automatically generate snow loads.

Step 1: Create a snow load case family

1. In the Pilot, right click **Loading** and select **Create a case family** from the context menu.
2. In the “Create a load case family [Solicitation]” dialog box, select **Snow EN 1991-1-3**.

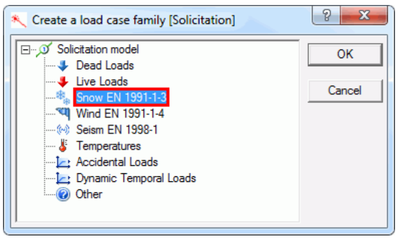


Figure 151: Selecting the snow load

3. Click **OK**.

A snow case family is created in the Pilot, in **Loading**, and a snow load case (**N**) is created by default.

Step 2: Define snow pressure according to the building's location

In this step, you will set the snow load case family properties.

1. In the Pilot, select the **Snow EN 1991-1-3** case family.
2. In the Properties window, set the properties of the **Snow EN 1991-1-3** case family:

   Parameters

   – In the “Snow load” field enter **0.52** kN/m².

   – In the “Altitude” field enter **250.00** m.

– From the **Exposure factor** drop down list select **Normal location**.

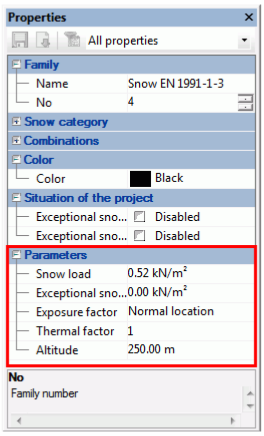


Figure 152: Setting the parameters of the snow loads

Step 3: Automatically generate snow loads

With the previously made settings, generate the snow loads automatically.

In the Pilot, right click **Snow EN 1991-1-3** case family and select **Automatic generation** from the context menu.

The snow loads are automatically generated.

Press <**ALT + 1**> for a better view of the loads. Right click the work area and select **Loads auto-scale**.

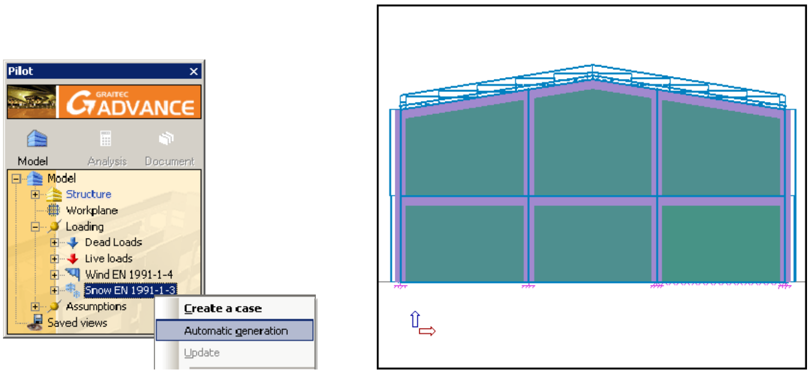


Figure 153: Automatic generation of the snow loads

Lesson 7: Defining the modal and seismic analysis

In this lesson you will define the modal and seismic analysis according to the current seismic standards.

You will learn how to:

- Create a seismic load family.
- Define the seismic region and the corner period zone.
- Configure the modal analysis parameters.

Step 1: Create a seismic load family

In this step, you will create a seismic load case family.

1. In the Pilot, right click **Loading** and select **Create a case family** from the context menu.

2. In the “Create a load case family [Solicitation]" dialog box, select **Seism EN 1998-1**.

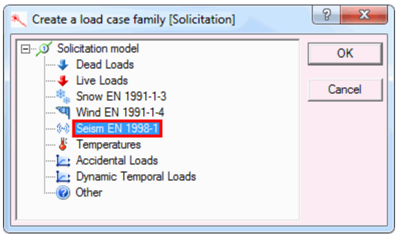


Figure 154: Selecting the **Seism EN 1998-1** case family

3. Click **OK**.

A seism case family and three load cases for each axis are created by default in the Pilot. Also, a modal analysis and the “Modes” case are generated.

Step 2: Define the seismic parameters

In the Pilot, select the **Seism EN 1998-1** case family.

In the Properties window make the following settings:

Implantation

– Enter a **0.08** agr/g value.

– From the “Soil type” drop-down list select **C**.

Structure

– Enter **1** as importance coefficient.

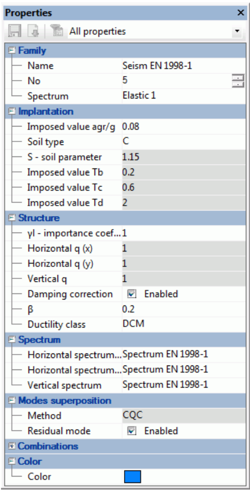


Figure 155: Seism case family properties

Viewing the horizontal response spectrum parameters in the "Function editor"

Spectrum

– Click **Horizontal spectrum**, then click to display the "Function editor" dialog box. The table on the left side lists the coordinates of the displayed spectrum.

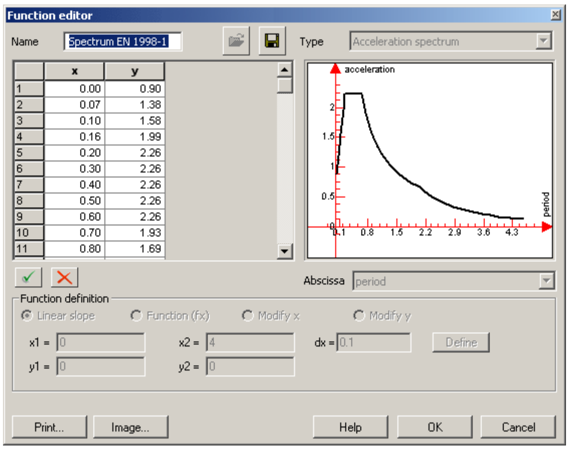


Figure 156: The "Function editor" for the horizontal definition

– Double click the graphic area to access the "Curves" dialog box that contains various display and configuration commands. The elastic response spectrum is displayed.

– Click to display the characteristic values.

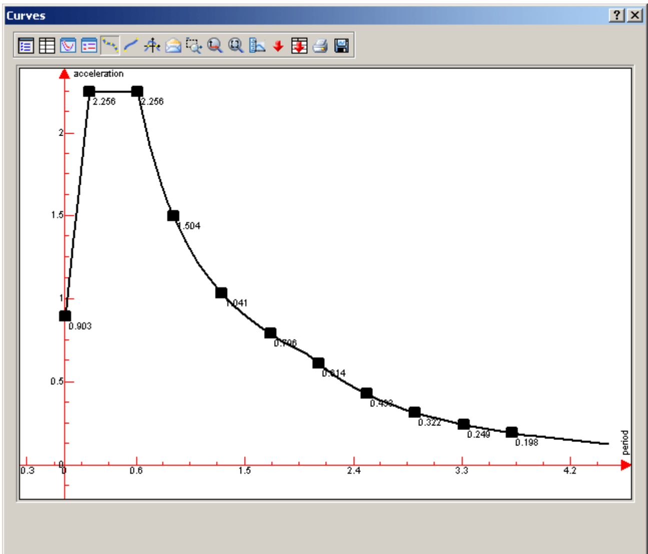


Figure 157: The intersection points of the abscissa

– Close the "Curves" dialog box and click **OK** in the "Function editor" dialog box.

Following the same process, it is possible to view the vertical response spectrum parameters.

Modes superposition

– From the "Method" drop-down list, in the seism Properties window, select **CQC**.

The seism cases are automatically generated for each of the three axes under **Seism EN 1998-1**. In order to obtain results only for the **EX** and **EY** cases, delete the **EZ** seism case from the Pilot.

Step 3: Configure the modal analysis parameters

The modal analysis and the “Modes” case are automatically generated at the creation of a seism case family.

1. In the Pilot, expand the **Assumptions > Modal analysis** and select **Modes**.

2. In the Properties window make the following settings:

<u>Modes</u>

– In the “Number” field enter the number of modes to calculate for the modal analysis: **25**.

<u>Masses</u>

– From the “Definition” drop-down list, select **Masses obtained by combining static loads**.

– Click **Combinations**, and then click .

In the “Combinations” dialog box, select the static load combinations to use for the modal analysis. From the “Available analyses” list:

✓Select the **SelfWeight** analysis, enter **1** in the “Coefficient” field and click **Add**.

✓Select the **LiveLoads** analysis, enter **0.4** in the “Coefficient” field and click **Add**.

✓Select the **Snw** analysis, enter **0.6** in the “Coefficient” field and click **Add**.

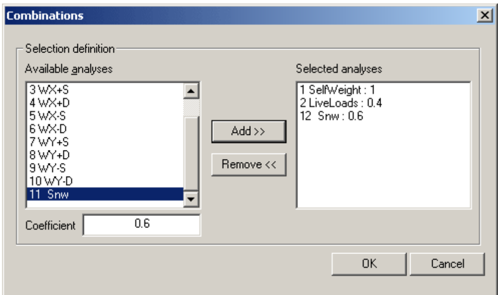


Figure 158: “Combinations” dialog box - selected analyses and coefficients

– Click **OK**.

– Enter **0%** in the “Percentage Z Dir.” field.

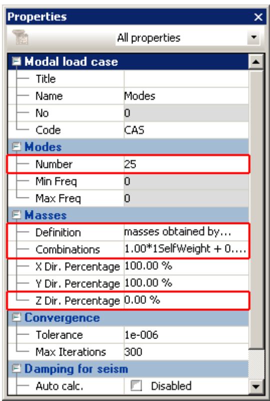


Figure 159: “Modes” Properties window

Lesson 8: Automatic creation of load combinations

In this lesson you will create load combinations according to the current standards.

You will learn how to:

- Create load combinations.
- View the combination properties.

Automatic generation of concrete and steel load combinations

1. In the Pilot, right click **Combinations** and select **Properties** from the context menu.

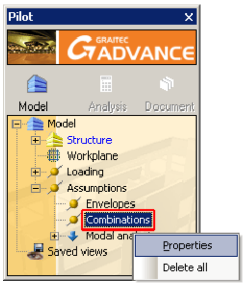


Figure 160: Accessing **Combinations** properties

The “Combinations” dialog box appears.

2. On the **Combinations** tab, click **Define**.
3. In the **Combinations options** dialog box, set “Live loads”, “Snow loads” and “Wind loads” as predominant actions, by enabling the corresponding check-boxes.

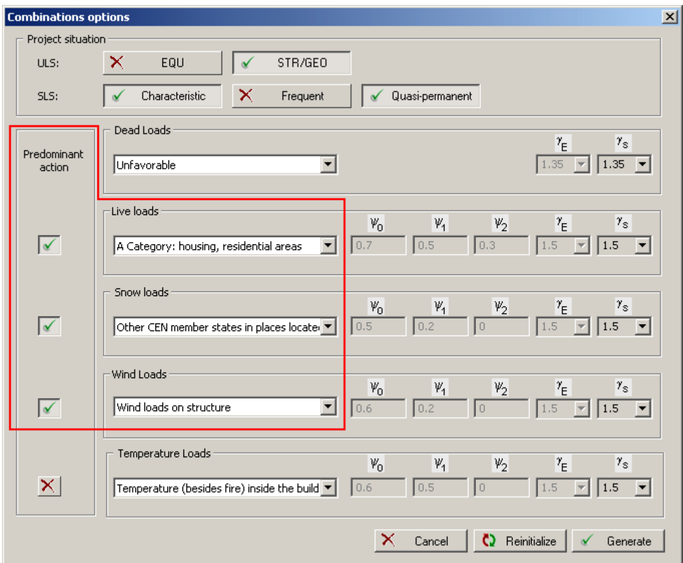


Figure 161: Generating combinations

4. Click **Generate**.
5. Click **Yes** in the message window to confirm the update of the load case codes.

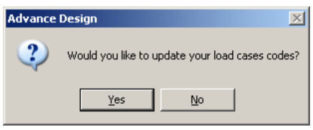


Figure 162: Confirming the update of the load cases

The **Combinations** tab displays all the load combinations.

Access the **Concrete** tab to view the load combinations for concrete.

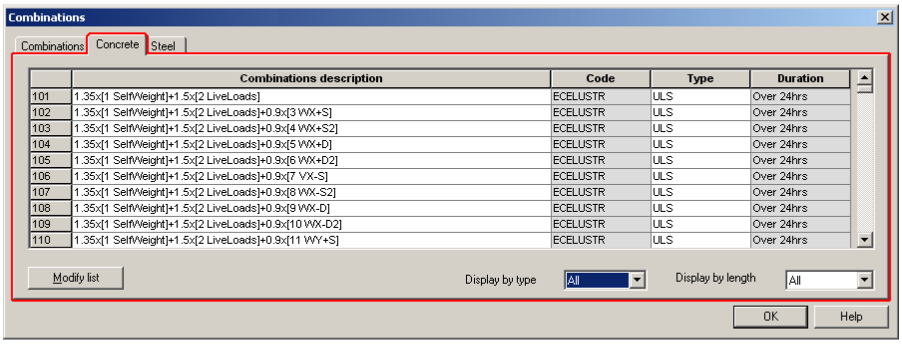


Figure 163: “Combinations” dialog box - **Concrete** tab

Access the **Steel** tab of the “Combinations” dialog box to view the load combinations for steel. The items are displayed by **Deflections** or by **Profiles**.

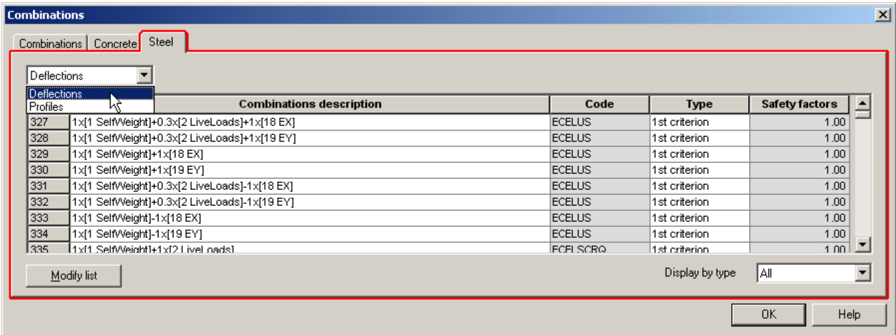


Figure 164: “Combinations” dialog box - **Steel** tab

The total number of combinations is displayed in the Pilot.

Figure 165: The number of load combinations

Lesson 9: Meshing and FE Calculation

The meshing is an essential operation for the structure calculation with the finite elements method. It performs a splitting-up of the structural elements in finite elements in order to refine the calculation results.

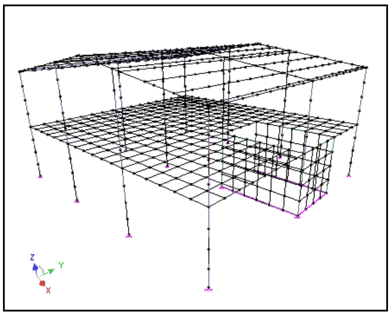


Figure 166: The nodes generated after the meshing

You will learn how to:

- Verify the descriptive model for errors.
- Configure the global mesh parameters: the mesh algorithm, the mesh elements size.
- Create the mesh.
- View the afferent linear loads that were acting initially on windwalls.
- Launch the finite elements calculation.

Step 1: Verify the descriptive model for errors

In this step, you will verify the descriptive model's consistency and accuracy using the **Verify** function.

1. On the **Modeling** toolbar click .
2. If the model is valid, the “No error found” message appears. Click **OK**.

Step 2: Set the mesh parameters

Next, you will configure the mesh algorithm and the mesh elements size. The elements will be split in quadrangular and triangular meshes.

1. From the main menu, select **Options > Mesh**.
2. In the “Mesh options” dialog box make the following settings:
   – From the “Mesh type” drop-down list, select **Grid**.
   – In the “Element type” area, select **Triangles and Quadrangles (T3-Q4)** to split the elements in triangular and quadrangular meshes with a node on each vertex.
   – Disable **Include loads in the mesh**. The loads will not affect the meshing of the elements.
   – Enter **0.8m** in the “Element size by default” field.

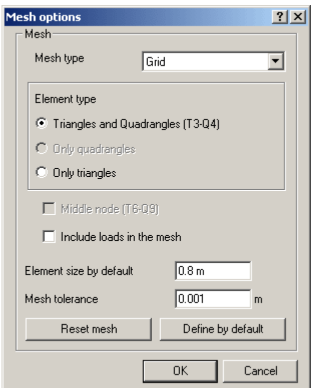


Figure 167: “Mesh options” dialog box

3. Click **OK**.

Step 3: Evaluate the model

In this step you will switch to the "Analysis" working mode to perform the meshing and verify the elements connectivity after the meshing, using the **Evaluate** function.

1. In the Pilot, click  to switch to the "Analysis" working mode.

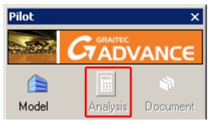


Figure 168: Pilot - **Analysis** button

2. In the "Calculation sequence" dialog box:

– Select the **Evaluate** option to verify the elements connectivity after the meshing. By selecting **Evaluate**, the verification and the meshing are automatically performed.

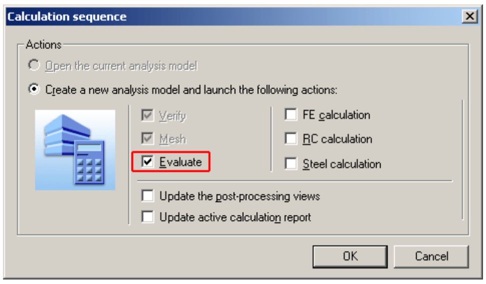


Figure 169: "Calculation sequence" dialog box

3. Click **OK**.

The model is verified for errors and the meshing is created. Also, the elements connectivity after the meshing is being verified. The command line displays information about each process as it is performed and the potential errors.

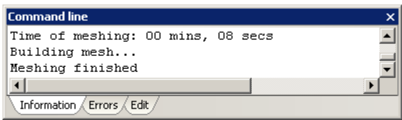


Figure 170: Processes information on the command line

When the process is completed, the meshed structure is displayed. The Pilot is in the "Analysis" mode, displaying the analysis model components. Also, the **Analysis - Assumptions** toolbar that groups the analysis configuration tools and commands is enabled.

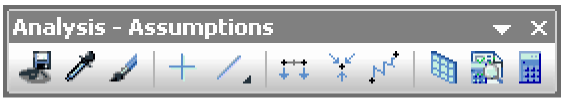


Figure 171: **Analysis - Assumptions** toolbar

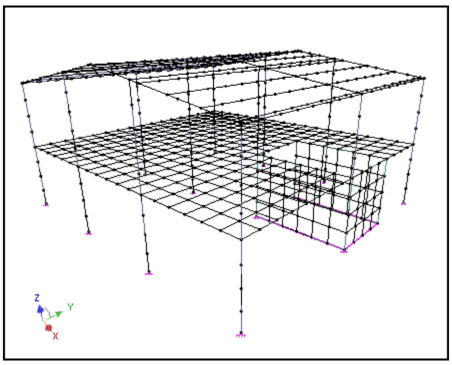


Figure 172: The nodes of the structure, generated after the meshing

Step 4: View the loads

1. Right click in the work area and select **Display > Loads** from the context menu.

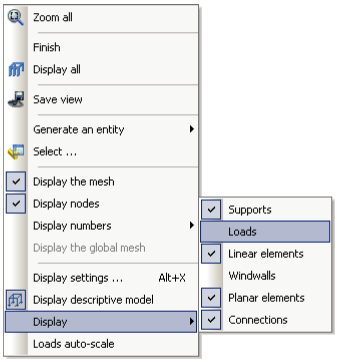


Figure 173: Displaying the loads from the work area context menu

2. Press <**ALT + R**> and rotate to view the loads.

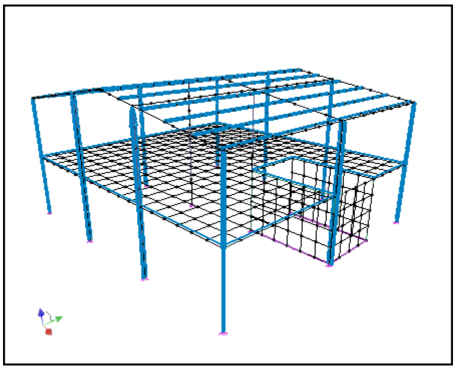
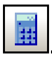


Figure 174: The loads on the meshed structure

Hide the loads: right click anywhere in the work area and unselect **Display > Loads**.

Step 5: Launch the finite elements calculation

After creating the analysis model and meshing the structure, the finite elements calculation can start.

1. On the **Analysis - Assumptions** toolbar, click .
2. In the “Calculation sequence” dialog box select the **Finite elements calculation** option.

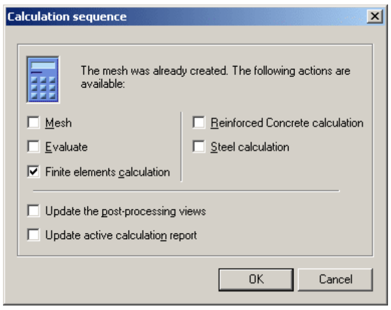


Figure 175: “Calculation sequence” dialog box

3. Click **OK**.

When the finite elements calculation is completed, the **Analysis - F.E. Results** toolbar appears.

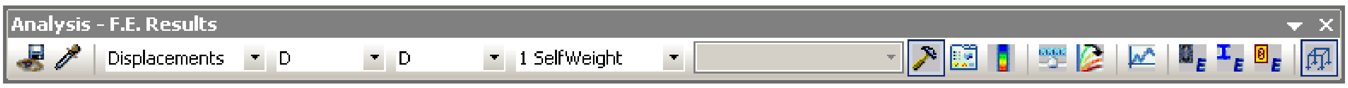


Figure 176: **Analysis - F.E. Results** toolbar

Lesson 10: Post-processing finite elements results

In this lesson you will test the results of the post-processing.

Advance Design provides a various set of options for the results display configuration during the post-processing step. These options allow you to select the analyses cases to process, the type of results, the results visualization mode in the graphic area, the elements to display with the analysis model during post-processing, etc.

You will learn how to:

- Display finite elements results on linear and planar elements.
- View enveloped results, eigen modes results and support reactions.
- Obtain the results diagram on a planar element using the section cut.
- View cross section stresses on a linear element.
- Display wall torsors.

Displaying F. E. results

Next, you will display finite elements results such as displacements and forces.

Displaying displacements of linear and planar elements

1. On the **Analysis - F. E. Results** toolbar:
   - Select the result type: **Displacements**.
   - From the “Available results for linear elements” drop-down list, select **D**.
   - From the “Available results for planar elements” drop-down list, select **D**.
   - Select the **101** combination from the analyses drop-down list.

Figure 177: **Analysis - F. E. Results** toolbar, analysis settings

2. Click  to start the post-processing.

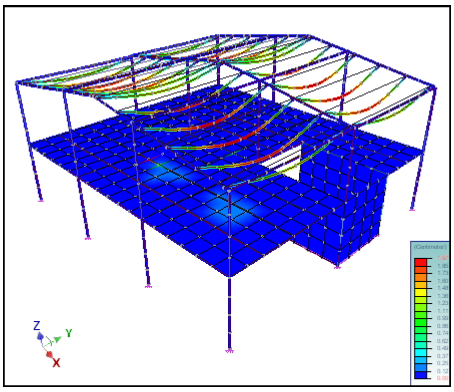


Figure 178: Displacements of the linear and planar elements of the structure

In the work area, the displacements results are displayed. The results are represented by colors, each color corresponding to a range of results. The results color legend is displayed at the bottom left corner of the window.

Note: *Keeping the <**Esc**> key pressed for a few seconds clears the graphical representation of the results in the work area.*

Displaying F. E. results on the linear elements of the structure

Displaying displacements on linear elements of the structure along the global Z-axis

1. On the **Analysis - F. E. Results** toolbar, from the "Available results for planar elements" drop-down list, select **None**: 

2. Click  to access the "Results" dialog box.

3. In the "Results" dialog box, on the **F.E.** tab, make the following settings:
   – Select the result type: **Displacements**.
   – Select results on **Linear elements**.
   – From the "Display mode" drop-down list select **Colors**.
   – Select **Dz** (displacement along the global **Z**-axis).

4. Click **OK**.

The post-processing starts.

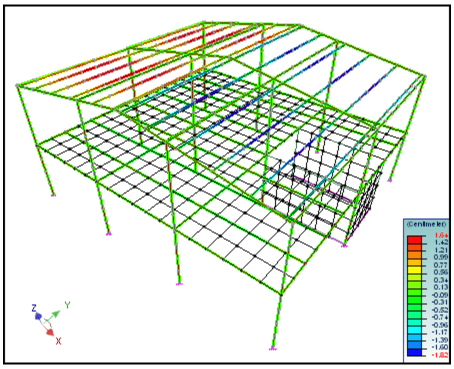


Figure 179: The displacements along the **Z**-axis on linear elements

Viewing displacements diagrams

First, you will display only the linear elements of the structure.

1. On the **Filters and selection** toolbar, click .

2. In the "Elements selection" dialog box, on the **Types** tab, expand **Entities > Model** and select **Linear**.

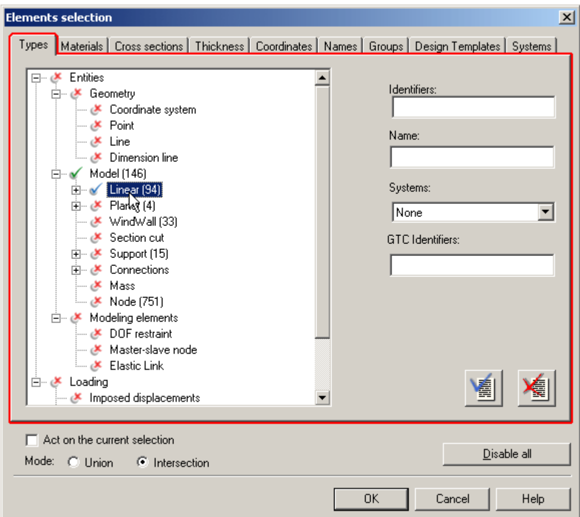


Figure 180: "Elements selection" dialog box – selecting linear elements

3. Click **OK**.

All the linear elements of the structure are selected.

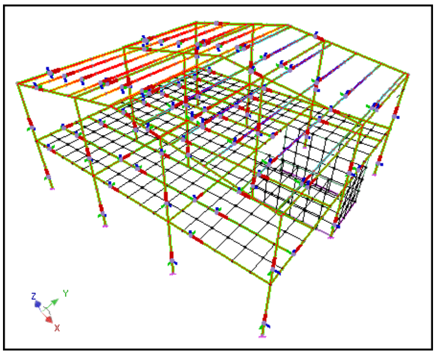


Figure 181: Linear elements of the structure

4. On the **Filters and selection** toolbar, click  to display only the selected elements of the structure (in this case the linear elements).

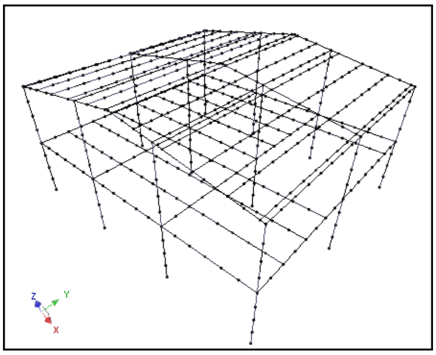


Figure 182: The linear elements of the structure isolated from the rest of the elements of the structure

Hide the nodes: right click anywhere in the work area and disable the **Display nodes** option from the context menu.

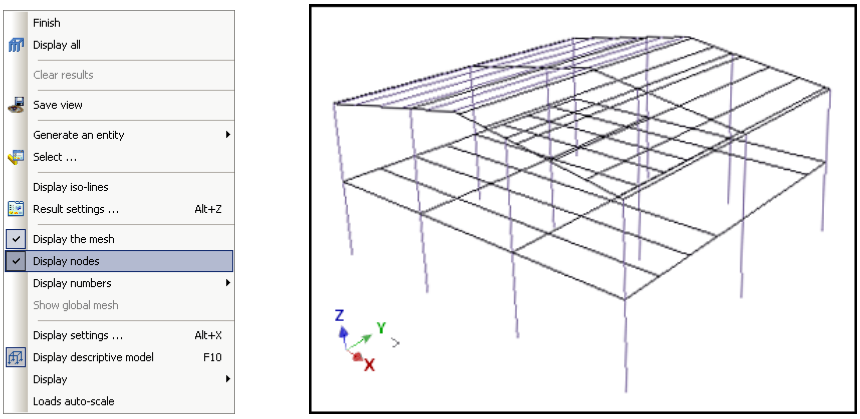


Figure 183: Hiding the nodes of the structure

Launch the post-processing:

1. On the **Analysis - F. E. Results** toolbar click .
2. In the “Results” dialog box, on the **F. E.** tab:
   – Select the **Displacements** analysis.
   – Select results on **Linear elements**.
   – From the “Display modes” drop-down list, select **Diagrams**.
3. Click **OK**.

The post-processing starts.

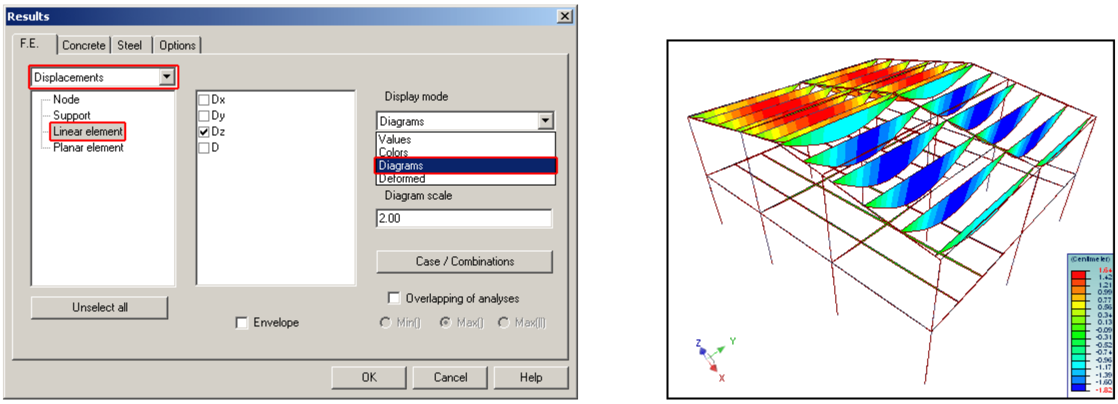


Figure 184: Diagrams of displacements

Press <**ALT + R**> and rotate to view the diagrams.

Press <**Esc**> to disable the rotating tool.

Displaying shear forces on the linear elements of the first portal frame along the local z-axis

Next, you will display forces on the linear elements of the first portal frame.

Display only the first portal frame of the structure.

Press <**ALT + 2**> for a right view.

On the **Predefined views** toolbar, click  to disable the perspective view.

1. Select the first portal frame as shown in Figure 185.

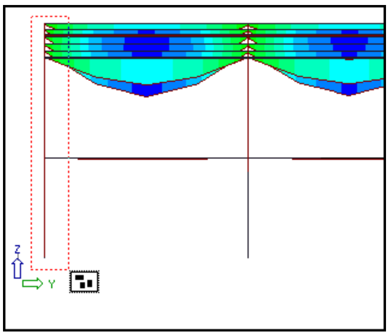


Figure 185: Selecting the first portal frame of the structure

2. On the **Filters and selection** toolbar, click  to display only the selected elements (the linear elements of the first portal frame).

Press <**ALT + 1**> for a front view.

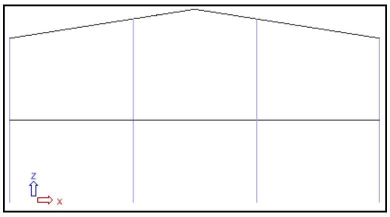


Figure 186: Front view of the first portal frame of the structure

Creating the post-processing of shear force along the local z-axis:

1. On the **Analysis - F. E. Results** toolbar, make the following settings:
   - Result type: **Forces**.
   - Results on linear elements: **Fz** (shear force along the local **z**-axis).
   - Results on planar elements: **None**.
   - Select the **101** case combination.

Figure 187: **Analysis - F. E. Results** toolbar, analysis settings

2. Click  to start the post-processing.

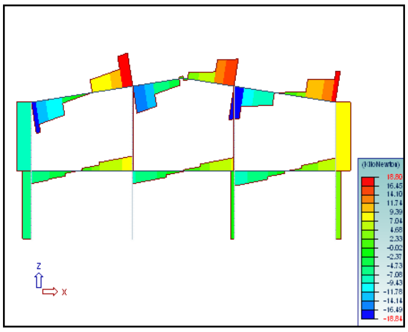


Figure 188: Forces on linear elements along the global **Z**-axis post-processing results

Displaying the values on diagrams

1. Press <**ALT + Z**> to display the "Results" dialog box.
2. On the **Options** tab, make the following settings:
   - In the "Values on diagrams" area, select the **Values on diagram** and **Display color of values** options.
   - In the "Font size" area, drag the slider to adjust the font size of the values.
3. Click **OK**.

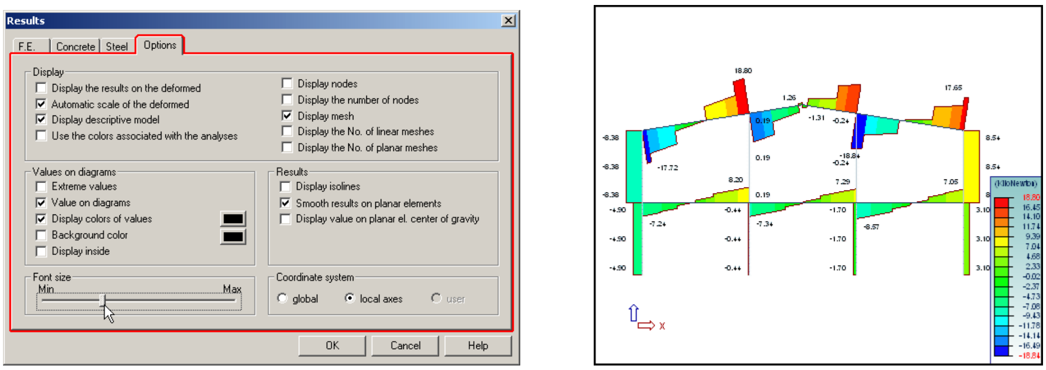


Figure 189: Displaying values on diagrams

Displaying the bending moment for the first portal frame

1. On the **Analysis - F. E. Results** toolbar, make the following settings:
   - Result type: **Forces**.
   - Results on linear elements: **My** (bending moment about the local **y**-axis).
   - Results on planar elements: **None**.
   - Select the **101** case combination.

Figure 190: **Analysis - F. E. Results** toolbar, analysis settings

2. Click  to start the post-processing.

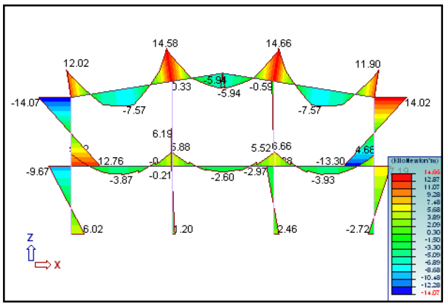


Figure 191: Bending moment on a portal frame post-processing result

Displaying F. E. results on the planar elements of the structure

In this step, you will display finite elements results on the planar elements of the structure.

Displaying bending moment about the local x-axis

Press <**ALT + R**> and rotate for a 3D view.

On the **Filters and selection** toolbar, click .

1. On the **Analysis - F.E. Results** toolbar, make the following settings:
   - Result type: **Forces**.
   - Results on linear elements: **None**.
   - Results on planar elements: **Mxx**.
   - Select the **101** case combination.

Figure 192: The **Analysis - F. E. Results**, analysis settings

2. Click  to start the post-processing.

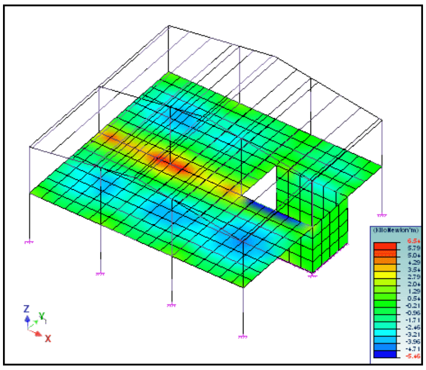
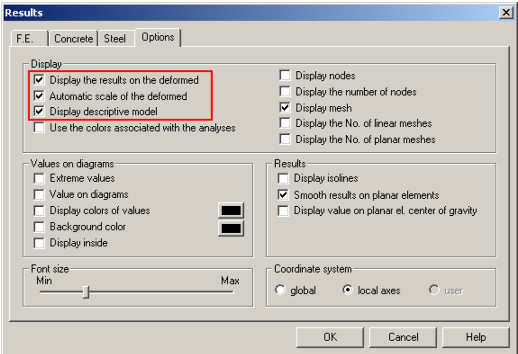


Figure 193: Bending moment about the local **x**-axis post-processing result

Displaying the results on the deformed shape

Next, you will view the result on the deformed plot.

1. On the **Analysis - F.E. Results** toolbar, click .
2. In the “Results” dialog box, on the **Options** tab, make the following settings:
   - In the “Display” area, select the **Display the result on the deformed**, **Automatic scale of the deformed** and **Display descriptive model** options.
   - In the “Values on diagrams” area, unselect all options.
3. Click **OK**. The post-processing starts.



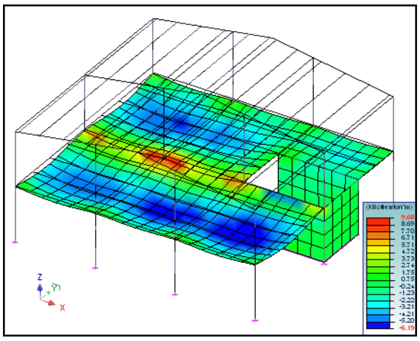


Figure 194: Results on the deformed shape

Viewing the animation for bending moment

1. Press <**ALT + 1**> for a front view.
2. On the **Analysis - F.E. Results** toolbar, click  to view the results in animation on the deformed elements.
3. Press <**Esc**> to end the animation.

Viewing the animation on the deformed shape

Press <**ALT + R**> and rotate for a 3D view.

1. On the **Analysis - F.E. Results** toolbar, click .
2. In the “Results” dialog box, on the **F. E.** tab, click the **Case / Combinations** button.
3. In the “Analyses and Combinations” dialog box, on the **Forces** tab:
   - Click **None** to unselect all the case combinations.
   - From the “Codes or identifiers” drop-down list, select **ECELSCRQ** to select all **ECELSCRQ** code combinations.

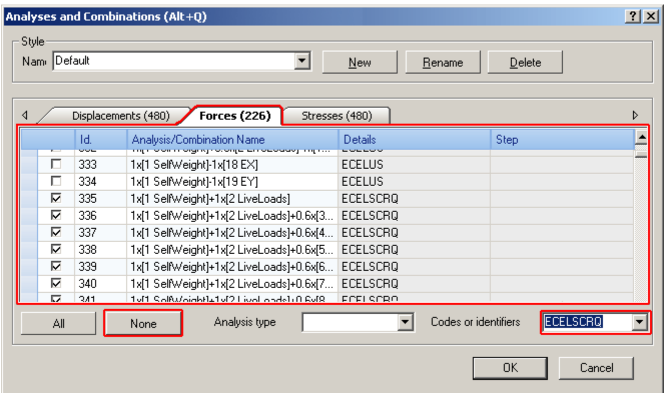


Figure 195: “Analyses and Combinations” dialog box, selecting the **ECELSCRQ** code combinations

4. Click **OK** to apply and close the dialog box.
5. In the “Results” dialog box, on the **F. E.** tab, select the **Envelope** option for **Max (II)** (envelope of the absolute value of the displacement).

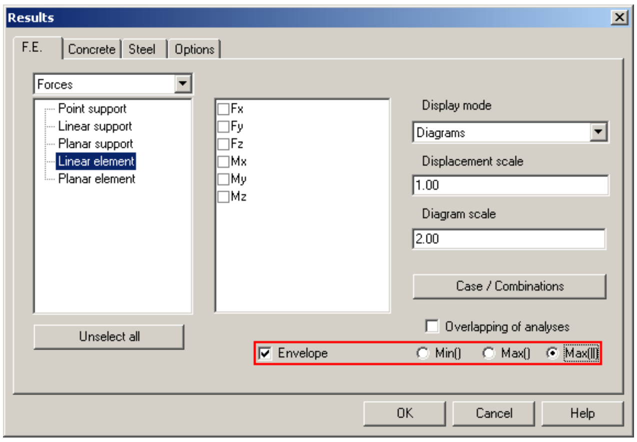


Figure 196: “Results” dialog box, selecting **Max (II)** envelope type

6. Click **OK**. The results are displayed.

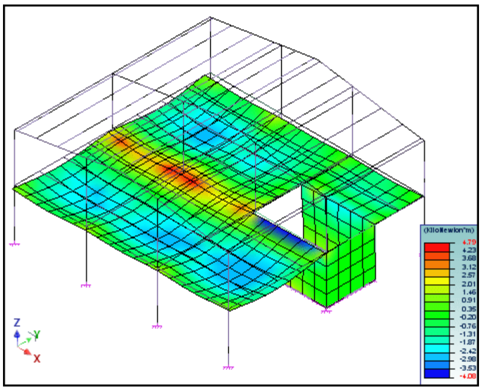


Figure 197: Deformed shape post-processing result for the **ECELSCRQ** case combinations

Use <**ALT + R**> and drag to rotate around the model and view the results. Press <**ALT + 5**> for a 3D view.

7. On the **Analysis - F.E. Results** toolbar, click  to view the results as animation.
8. Press <**Esc**> to end the animation.

Displaying the eigen mode 1 animation

1. On the **Analysis - F.E. Results** toolbar, make the following settings:

– Result type: **Eigen modes**.

– From the “Eigen modes” drop-down list, select **Eigen mode 1**.

Figure 198: **Analysis - F.E. Results** toolbar, analysis settings

2. Click  to start the post-processing.

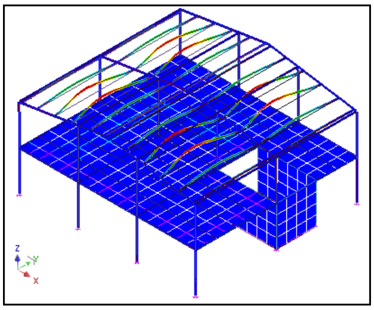


Figure 199: Results for eigen mode 1

3. On the **Analysis - F.E. Results** toolbar, click  to start the animation.

4. Press <**Esc**> to end the animation.

Displaying the eigen mode 5 animation

1. On the **Analysis - F.E. Results** toolbar, make the following settings:

– Result type: **Eigen modes**.

– From the “Eigen modes” drop-down list, select **Eigen mode 5**.

Figure 200: **Analysis - F.E. Results** toolbar, analysis settings

2. Click  to start the post-processing.

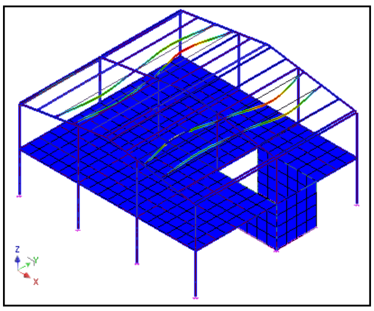


Figure 201: Results for the eigen mode 5

3. On the **Analysis - F.E. Results** toolbar, click  to start the animation.

4. Press <**Esc**> to end the animation.

Displaying forces results on the point supports

1. On the **Analysis - F.E. Results** toolbar: click .
2. In the “Results” dialog box, on the **F. E.** tab, make the following settings:
   – Select the result type: **Forces**.
   – Unselect all the results: click **Unselect all**.
   – From the elements list, select **Point support**.
   – Select **Fz**.

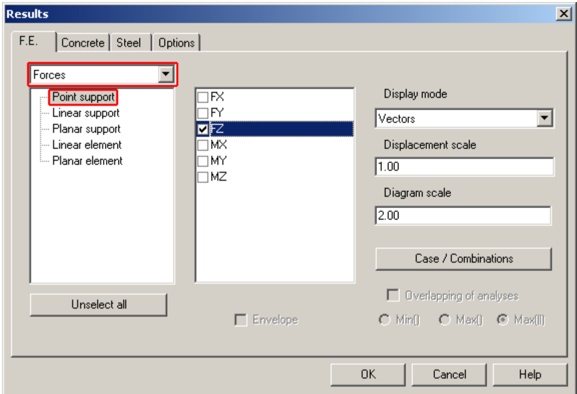


Figure 202: “Results” dialog box - selecting **Forces** type result for supports on the global **Z**-axis

   – Click **Case / Combinations**. On the **Forces** tab, select **ECELUS** from the “Codes or identifiers” drop-down list.
3. Click **OK** to apply and close the “Analyses and Combinations” dialog box.
4. In the “Results” dialog box, click **OK**. The post-processing starts.

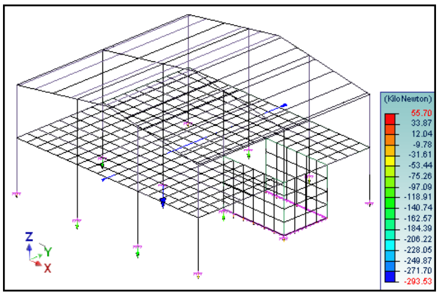


Figure 203: Forces on supports post-processing result

Displaying the results on the point supports of the first portal frame

1. Select the four supports of the first portal frame as in Figure 204.

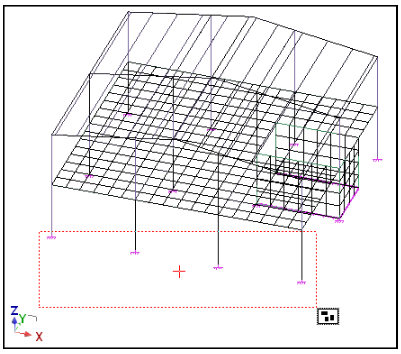


Figure 204: Selecting the supports of the first portal frame

2. Click  to start the post-processing.

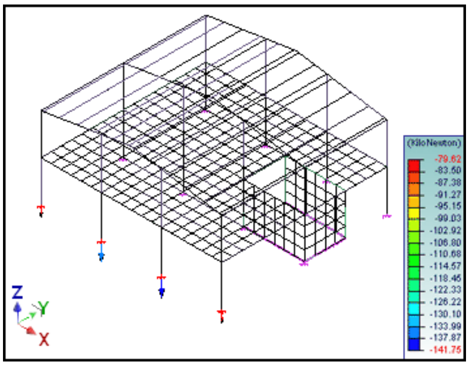


Figure 205: Forces on supports of the first portal frame post-processing result

Displaying values on the diagrams

1. On **Analysis - F.E. Results** toolbar, click .
2. In “Results” dialog box, on the **Options** tab, select the **Value on diagrams** and **Display colors of values** options.

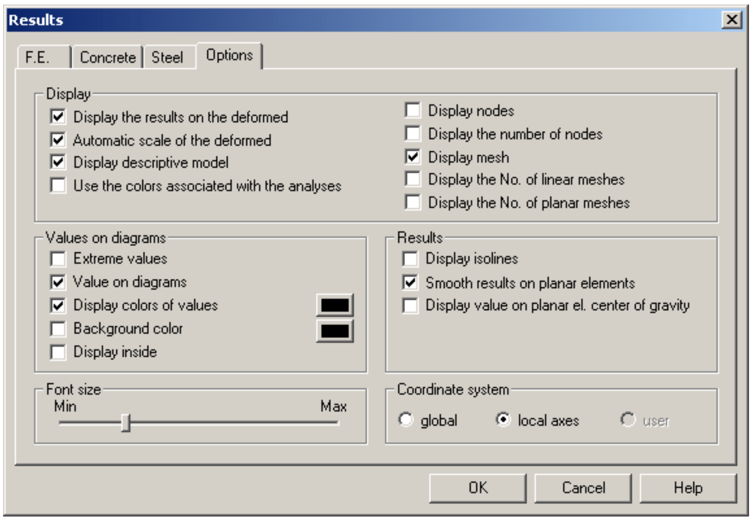


Figure 206: “Results” dialog box - display settings

3. Click **OK**.

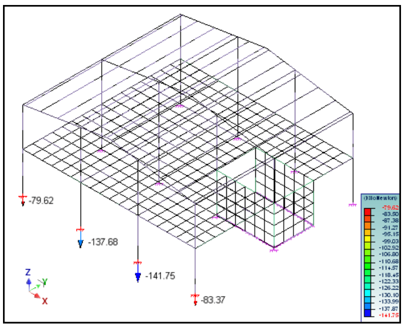


Figure 207: Displaying values on diagrams

4. Press <**Esc**> to unselect the supports and clear the graphical display of the results.

Obtaining the results diagram on a planar element using the section cut

1. From the main menu, select **Generate > Section cut**.

2. Create a section cut line on the planar element (floor), as in Figure 208.

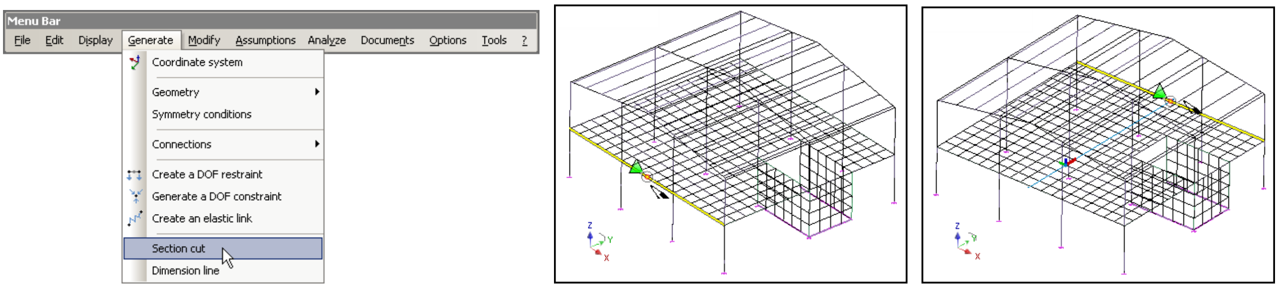


Figure 208: Generating the section cut

Make sure that the "Midpoint" snap mode is enabled.

3. In the work area, click the section cut to select it.

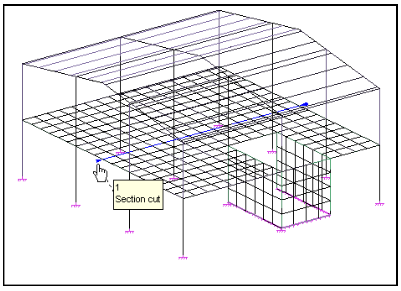
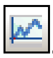
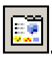


Figure 209: Selecting the section cut

To configure the content of the information dialog box that appears each time the mouse cursor is above an element, double click  *on the Status bar.*

4. On the **Analysis - F.E. Results** toolbar, click . The "Result curves" dialog box appears.

- In the "Result curves" dialog box, click .
- In the "Curves" dialog box, click the **Case / combinations** button.

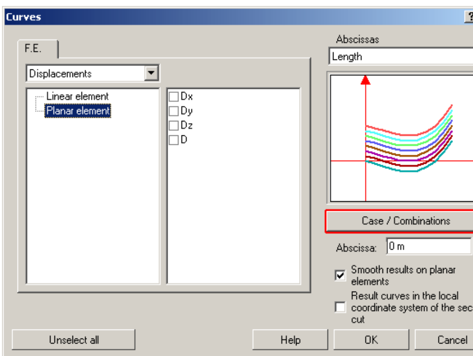


Figure 210: "Curves" dialog box

- The "Analyses and Combinations" dialog box appears. On the **Displacements** tab click **None** to unselect all combinations, then select **ECELSCRQ** from the "Codes or identifiers" drop-down list.
- Click **OK** to close the "Analysis and Combinations" dialog box.

5. Click **OK** to close the “Curves” dialog box.

6. Select the first graphic frame of the “Result curves” dialog box.

7. Select the **Displacements** result type on **D**.

8. Click  to start the post-processing.

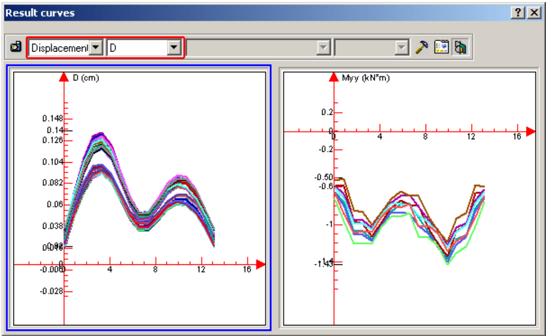


Figure 211: “Result curves” dialog box - Displacements

9. Double click the first graphic frame. The “Curves” dialog box appears. Click  and use the ruler to click anywhere on the graphic. The “Coordinates” dialog box displays the abscissa position and the results of the corresponding points.

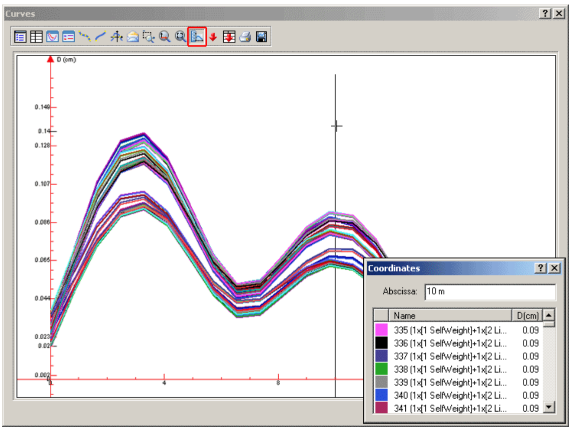


Figure 212: ECELSCRQ results on specified coordinates

10. Close the “Coordinates” dialog box.

Viewing the minimum and maximum envelopes values.

1. In the “Curves” dialog box, click  and on the graph. You can see the minimum and maximum envelope values.

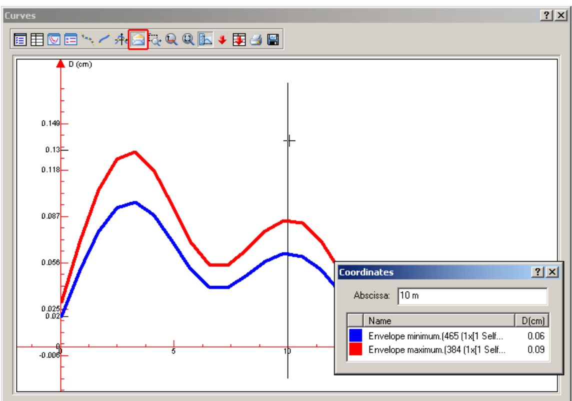


Figure 213: Envelope results on specified coordinates

2. Close the envelopes dialog box.

3. Click  to display the curve points and their result values.

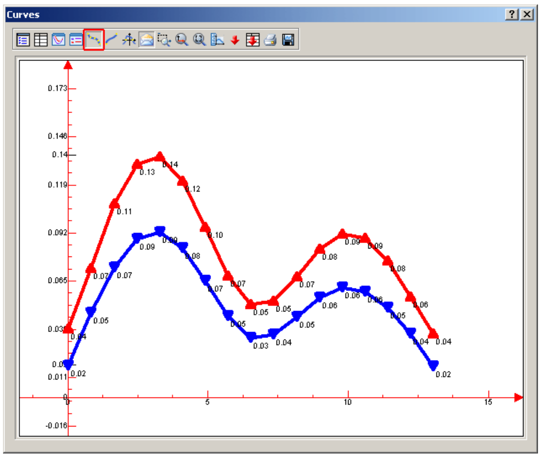


Figure 214: Curve points

Viewing the cross section stresses on the length of a column

Before starting: right click anywhere in the work area and select **Cancel selection** from the context menu, to unselect all the elements of the model.

1. Select a column of **GroundFloor**, as in Figure 215.

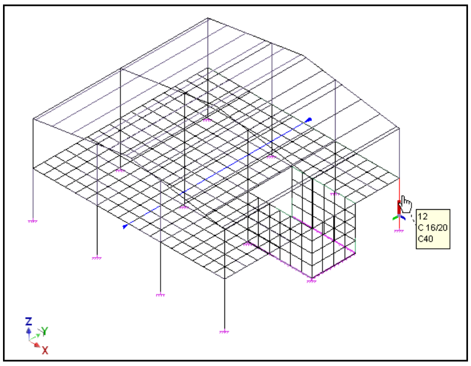


Figure 215: Selecting a column

2. Right click anywhere in the work area and select **Cross section stresses** from the context menu.

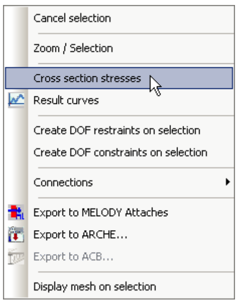


Figure 216: Accessing the “Linear elements stresses” dialog box

3. In the "Linear elements stresses" dialog box, click the **Case / Combinations** button.

4. On the **Stresses** tab, click **None** to unselect all combinations, then select **ECELSCRQ** from the “Codes or identifiers” drop-down list.

5. Click **OK**.

The analysis of the stresses distribution is displayed on the cross section of the column. Drag the slider to specify the abscissa on the element length on which you want to view the stresses distribution. The stresses diagram displays the results corresponding to the specified point on the element.

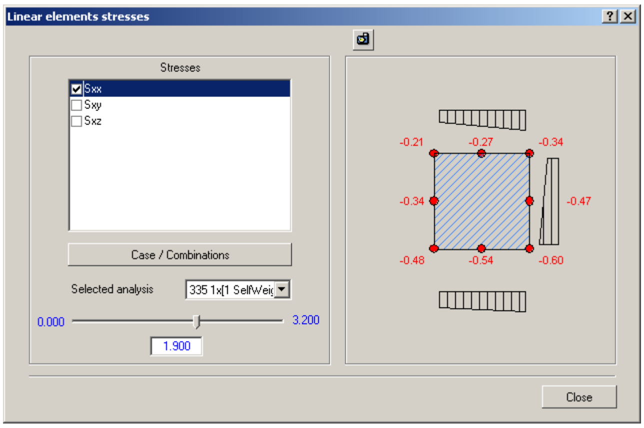


Figure 217: "Linear elements stresses" dialog box

Viewing torsors on a wall

The torsors allow the visualization of the results as diagrams, for the "wall" planar elements (planar elements parallel to the global **Z**-axis).

Torsors of the moment in the wall's plane

1. After unselecting all the elements, select a wall, as in Figure 218.

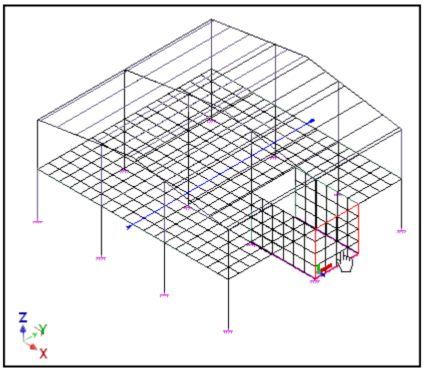


Figure 218: Selecting a wall

On the **Filters and selection** toolbar, click  to only display the selected wall.

Double click the scroll mouse button to zoom on the displayed element.

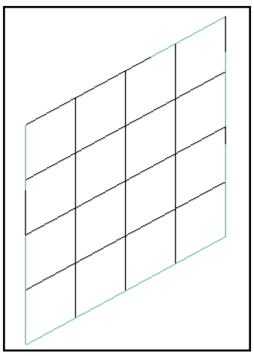


Figure 219: Isolated wall

2. On the **Analysis - F. E. Results** toolbar, make the following settings:

- Result type: **Torsors**.
- Results on planar elements: **Mz.**
- Select the **335** case combination.

Figure 220: **Analysis - F.E. Results** toolbar, analysis settings

3. Click  to start the post-processing.

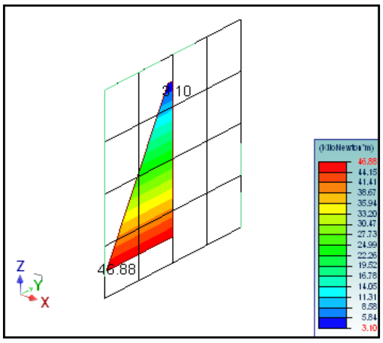


Figure 221: Torsors of the moment in the wall's plane post-processing result

Using the same process it is possible to view several more wall results: M (moment in the element's plane), T (Shear force in the element's plane), etc.

Display the structure elements:

1. On the **Filters and selection** toolbar, click .

2. Double click the scroll mouse button for a fit on screen view of the structure.

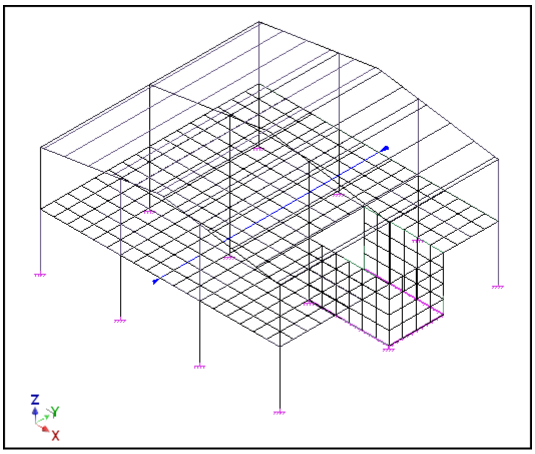
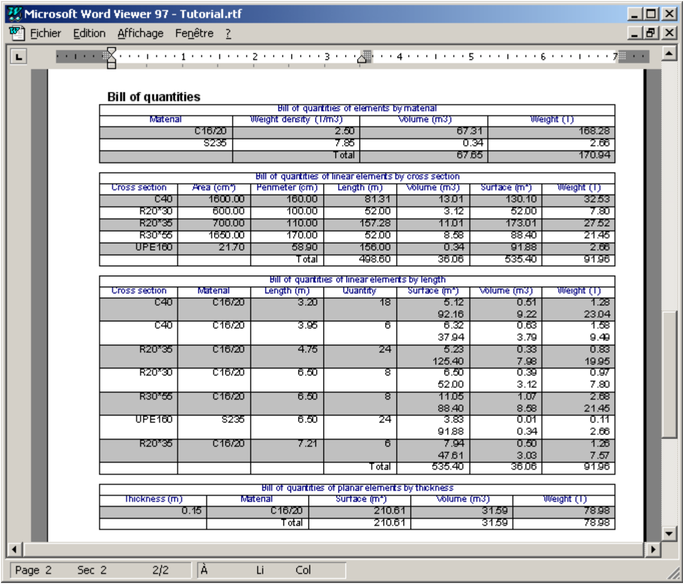


Figure 222: The structure in "Axes" rendering mode

Lesson 11: Creating reports and post-processing views

With Advance Design you can generate complete calculation reports, containing data and results in different outputs (tables, texts, images, graphical post-processing, etc.).

In this lesson you will generate a bill of quantities report, a report for linear elements efforts by load case, a report for the characteristic values of eigen modes and a report of the efforts on the linear elements of the front of the structure. During the generation of the reports, you will also create the cover sheets, save views of the results to add them in the documents and configure the contents of the documents.



Bill of quantities

Bill of quantities of elements by material			
Material	Weight density (T/m3)	Volume (m3)	Weight (T)
C16/20	2.50	67.31	168.28
S235	7.85	0.34	2.66
	Total	67.65	170.94

Bill of quantities of linear elements by cross section						
Cross section	Area (cm²)	Perimeter (cm)	Length (m)	Volume (m3)	Surface (m²)	Weight (T)
C40	1600.00	160.00	81.31	13.01	130.10	32.53
R20*30	600.00	100.00	52.00	3.12	52.00	7.80
R20*35	700.00	110.00	157.28	11.01	173.01	27.52
R30*55	1650.00	170.00	52.00	8.58	88.40	21.45
UPE160	21.70	58.90	156.00	0.34	91.88	2.66
		Total	498.60	36.06	535.40	91.96

Bill of quantities of linear elements by length						
Cross section	Material	Length (m)	Quantity	Surface (m²)	Volume (m3)	Weight (T)
C40	C16/20	3.20	18	5.12 92.16	0.51 9.22	1.28 23.04
C40	C16/20	3.95	6	6.32 37.94	0.63 3.79	1.58 9.49
R20*35	C16/20	4.75	24	5.23 125.40	0.33 7.98	0.83 19.95
R20*30	C16/20	6.50	8	6.50 52.00	0.39 3.12	0.97 7.80
R30*55	C16/20	6.50	8	11.05 88.40	1.07 8.58	2.68 21.45
UPE160	S235	6.50	24	3.83 91.88	0.01 0.34	0.11 2.66
R20*35	C16/20	7.21	6	7.94 47.61	0.50 3.03	1.26 7.57
			Total	535.40	36.06	91.96

Bill of quantities of planar elements by thickness				
Thickness (m)	Material	Surface (m²)	Volume (m3)	Weight (T)
0.15	C16/20	210.61	31.59	78.98
	Total	210.61	31.59	78.98

Figure 223: “Bill of quantities” report file

You will learn how to:

- Configure and generate a report.
- Save post-processing views.
- Insert post-processing views in the report.
- Generate a report.
- Generate a report of filtered elements and results.

Creating a bill of quantities

In this step you will create the “Bill of quantities” report. For an easier generation of reports, Advance Design provides a set of report templates, with a predefined configuration of tables and data. The bill of quantities report allows you to view quantitative details of the structure (materials, cross sections, lengths and thicknesses).

1. From the main menu, select **Documents > Bill of quantities**. The “Report generator” dialog box appears. On the left side, the content of the report is displayed. By default, the “Bill of quantities” report contains the “Bill of quantities of elements by material”, “Bill of quantities of linear elements by cross section”, “Bill of quantities of linear elements by length” and “Bill of quantities of planar elements by thickness” tables.

2. On the left side of the "Report generator" dialog box, right click **Cover sheet** and select **Properties**.

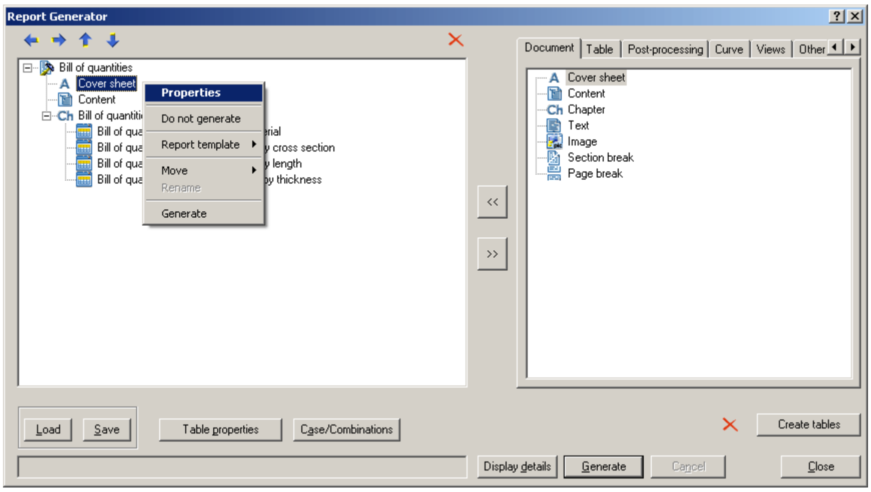


Figure 224: "Report generator" dialog box, selecting **Cover sheet** properties

3. In the "Cover sheet" dialog box, make the following settings:

– In the "Document type" field, enter **Bill of quantities**.

– In the "N°" field enter the ID number **01**.

– In the "Author" field, enter the author name.

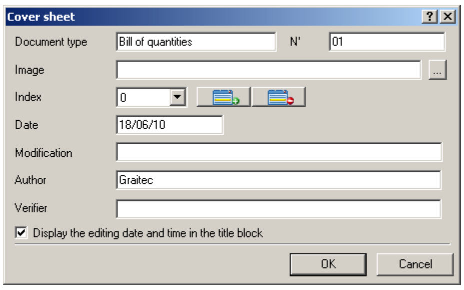
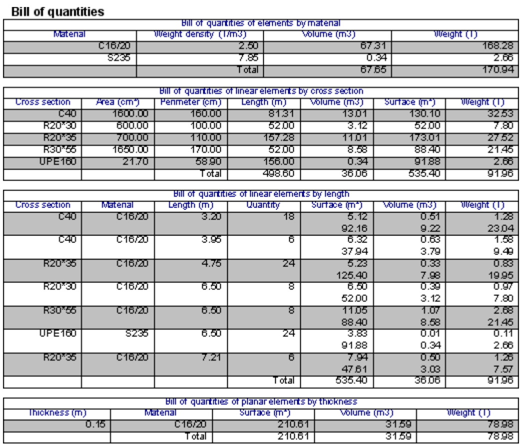


Figure 225: "Cover sheet" dialog box

4. Click **OK**.

5. In the "Report generator" dialog box click **Generate**.

The "Bill of quantities" report file opens. The cover sheet contains the information about the report. On the second page there is the "Bill of quantities" report. It contains the "Bill of quantities of elements by material", "Bill of quantities of linear elements by cross section", "Bill of quantities of linear elements by length" and "Bill of quantities of planar elements by thickness" tables. In each table there are the attributes of the elements.



Bill of quantities

Bill of quantities of elements by material			
Material	Weight density (T/m3)	Volume (m3)	Weight (T)
C16/20	2.50	67.31	168.28
S235	7.85	0.34	2.66
	Total	67.65	170.94

Bill of quantities of linear elements by cross section						
Cross section	Area (cm²)	Perimeter (cm)	Length (m)	Volume (m3)	Surface (m²)	Weight (T)
C40	1600.00	160.00	81.31	13.01	130.10	32.53
R20*30	600.00	100.00	52.00	3.12	52.00	7.80
R20*35	700.00	110.00	157.28	11.01	173.01	27.52
R30*55	1650.00	170.00	52.00	8.58	88.40	21.45
UPE160	21.70	58.90	156.00	0.34	91.88	2.66
		Total	498.60	36.06	535.40	91.96

Bill of quantities of linear elements by length						
Cross section	Material	Length (m)	Quantity	Surface (m²)	Volume (m3)	Weight (T)
C40	C16/20	3.20	18	5.12 92.16	0.51 9.22	1.28 23.04
C40	C16/20	3.95	6	6.32 37.94	0.63 3.79	1.58 9.49
R20*35	C16/20	4.75	24	5.23 125.40	0.33 7.98	0.83 19.95
R20*30	C16/20	6.50	8	6.50 52.00	0.39 3.12	0.97 7.80
R30*55	C16/20	6.50	8	11.05 88.40	1.07 8.58	2.68 21.45
UPE160	S235	6.50	24	3.83 91.88	0.01 0.34	0.11 2.66
R20*35	C16/20	7.21	6	7.94 47.61	0.50 3.03	1.26 7.57
			Total	535.40	36.06	91.96

Bill of quantities of planar elements by thickness				
Thickness (m)	Material	Surface (m²)	Volume (m3)	Weight (T)
0.15	C16/20	210.61	31.59	78.98
	Total	210.61	31.59	78.98

Figure 226: The "Bill of quantities" .rtf report file

Creating a report with filtered results

In this step, you will create a report with results on linear elements.

1. In the "Report generator" dialog box:

   – Click to empty the list. Click **Yes** in the confirmation dialog box.

   – Select the **Table** tab.

   – Select the result table: **Finite Elements Analysis > Results > Forces > Linear elements**.

   – Select **Linear elements forces by load case** and click to add it in the Document structure.

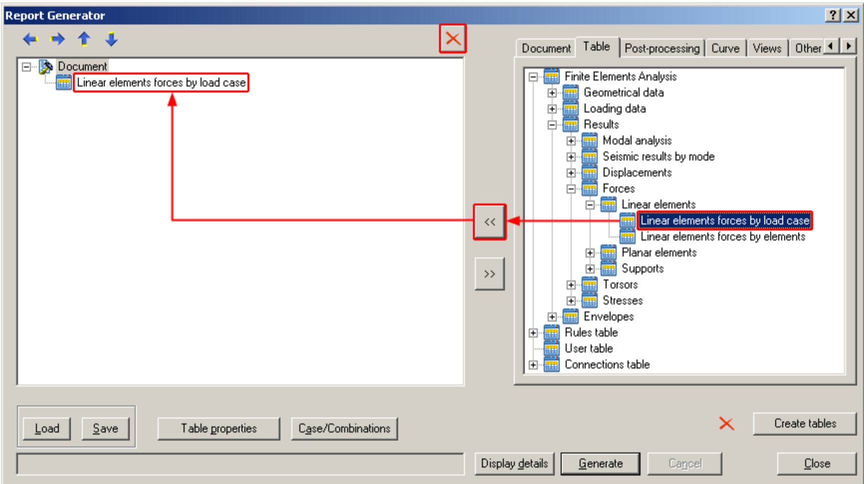


Figure 227: "Report Generator" dialog box - adding **Linear elements efforts by load case** in the Document structure

2. In the Document structure, right click **Linear elements forces by load case** and select **Properties**.

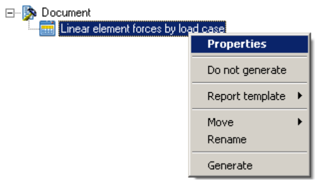


Figure 228: "Report Generator" dialog box, selecting **Linear elements forces by load case** properties

3. The "Table properties" dialog box appears. In this dialog, make the following settings:

   – In the "Results options" area, from the "Results on" drop-down list, select **Endpoints and middle (super element)**.

   – Click **Advanced options** to display the "Specific selection" panel.

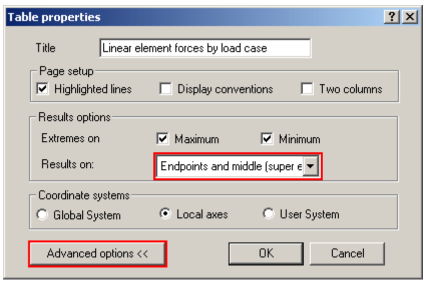


Figure 229: "Table properties" dialog box

4. On the “Specific selection” panel, make the following settings:
   - Select the **Define a specific selection of loads cases / combinations** option.
   - From the “Analysis type” drop-down list, select **None**.
   - Select the **327** and **328** load cases combinations.

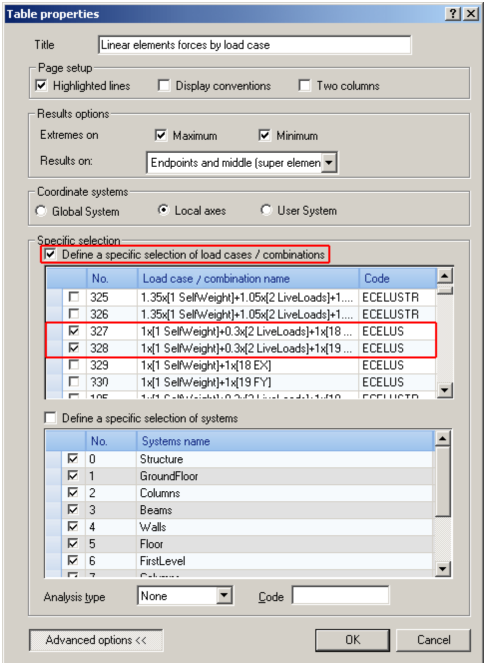


Figure 230: “Table properties” dialog box, setting the report contents

5. Click **OK**.
6. In the “Report generator” click **Generate**.

The document in .rtf format opens. It contains the "Linear elements forces by load case” table, with the efforts and the moments for each axis, calculated for the **327** and **328** case combinations.

Generating a report with eigen modes results

For modal analysis you can choose to view the eigen modes results on linear and planar elements. The eigen modes results are available only in the "deformed" visualization mode.

Saving Eigen modes post-processing views

Display the results of Eigen mode 1, Eigen mode 2 and Eigen mode 3 and save the post-processing views to add them in the report.

Save the **Eigen mode 1** post-processing view.

Press <**ALT + 5**> for a 3D view.

1. On the **Analysis - F.E. Results** toolbar, make the following settings:
   - Result type: **Eigen modes**.
   - From the list of **Eigen modes** analysis, select **Eigen mode 1**.

Figure 231: **Analysis - F.E. Results** toolbar, analysis settings

2. Click  to start the post-processing.

3. On the **Analysis - F.E. Results** toolbar, click to save the post-processing view.

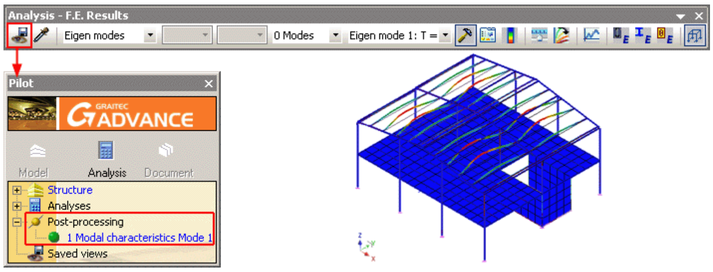


Figure 232: Saving a post-processing view

The post-processing view is saved in the Pilot, in “Post-processing”.

<u>Save the **Eigen mode 2** post-processing view</u>:

1. On the **Analysis - F.E. Results** toolbar, make the following settings:

– Result type: **Eigen modes**.

– From the list of **Eigen modes** analysis, select **Eigen mode 2**.

Figure 233: **Analysis - F.E. Results** toolbar, analysis settings

2. Click to start the post-processing.

3. Press <**ALT + 3**> for a top view.

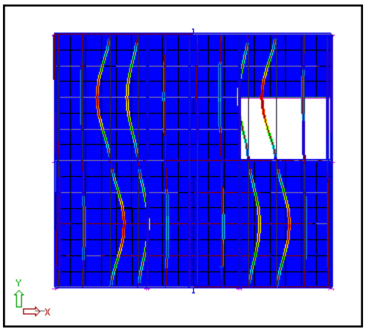


Figure 234: Eigen mode 2 post-processing result

4. On the **Analysis - F.E. Results** toolbar, click to save the post-processing view.

<u>Save the **Eigen mode 3** post-processing view</u>:

1. On the **Analysis - F.E. Results** toolbar, make the following settings:

– Result type: **Eigen modes**.

– From the list of **Eigen modes** analysis, select **Eigen mode 3**.

Figure 235: **Analysis - F.E. Results** toolbar, analysis settings

2. Click to start the post-processing.

3. Press <**ALT + 3**> for a top view.

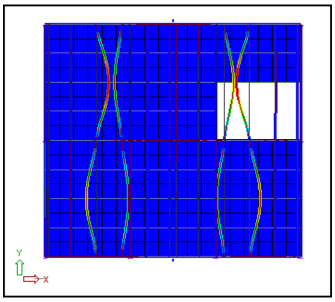


Figure 236: Eigen mode 3 post-processing result

4. On the **Analysis - F.E. Results** toolbar, click to save the post-processing view.

The three views are saved in the Pilot, in Post-processing.

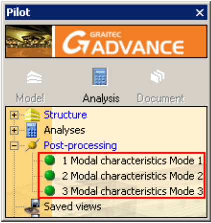


Figure 237: Modal characteristics modes are saved in the Pilot

Generating a report for the characteristic values of eigen modes

1. From the main menu, select **Documents > Generate a new report**.

2. In the “Report generator” dialog box:

- Click to empty the list.
- On the **Table** tab, select **Finite Elements Analysis > Results > Modal analysis > Characteristic values of eigen modes** and click to add it in the Document structure.
- On the **Post-processing** tab, select the **1 Modal characteristics Mode 1** view, and add it in the Document structure.

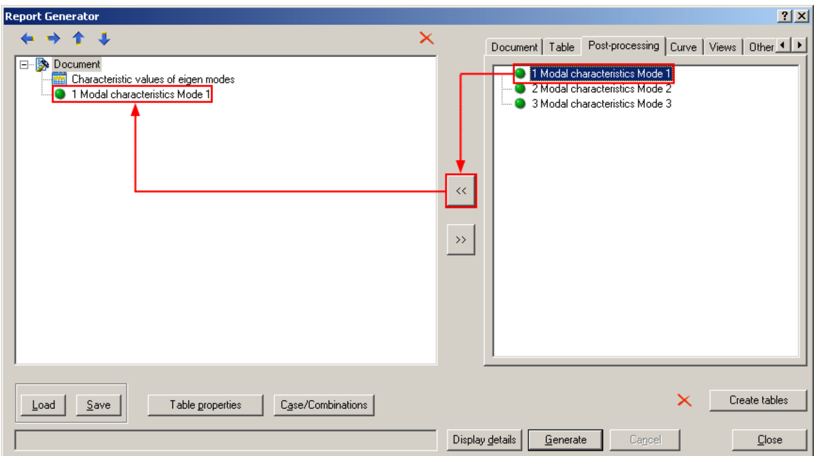


Figure 238: “Report generator” dialog box, adding the **Modal characteristics Mode 1** in the Document structure

Double clicking an item from the available data adds it to the document list.

- Double click the other two views to add them in the Document structure.

3. Click **Generate**.

The .rtf report file opens. It contains the "Characteristic values of eigen modes" table: "Mode", "Pulsation", "Period", "Frequency", "Energy", "Modal masses" and "Damping" values. It also contains the saved views.

Generating a report with linear element efforts result

Next, you will generate a report containing effort results for the first portal frame of the structure, including the saved post-processing views of these results.

Saving forces post-processing views

Save the post-processing view for forces along the local **X**-axis.

1. Press <**ALT + 2**> for a right view. Select the first portal frame.

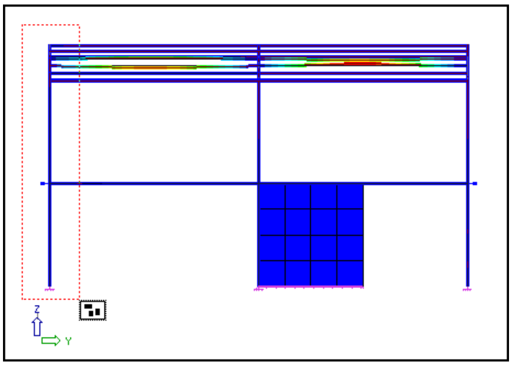


Figure 239: Selection of the first portal frame

2. On the **Filters and selection** toolbar, click  to display only the selected elements.

Press <**ALT + 1**> for a front view.

3. On the **Analysis - F. E. Results** toolbar, make the following settings:
   - Result type: **Forces**.
   - Results on linear elements: **Fx** (force along the local **x**-axis).
   - Results on planar elements: **None**.
   - Select the **327** case combination.

Figure 240: **Analysis - F. E. Results** toolbar , analysis settings

4. Click  to start post-processing.

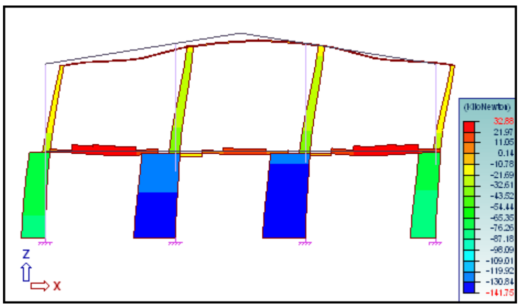


Figure 241: Efforts along the local **x**-axis post-processing result

5. Press <**ALT + Z**> to display the "Results" dialog box.

6. Select the **Options** tab and make the following settings:
   - In the “Values on diagrams” area, select the **Value on diagrams** option.
   - In the “Display” area, disable the **Display the results on the deformed** option.

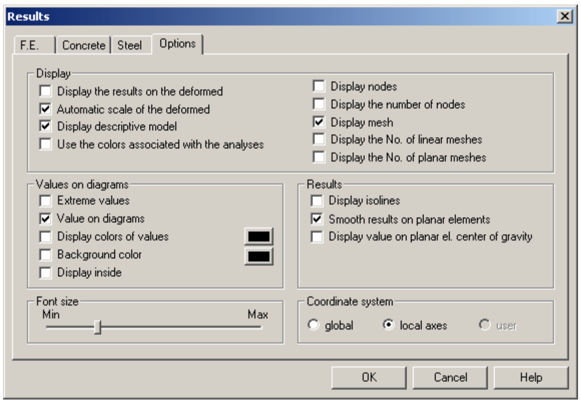


Figure 242: “Results” dialog box, display settings

7. Click **OK**.

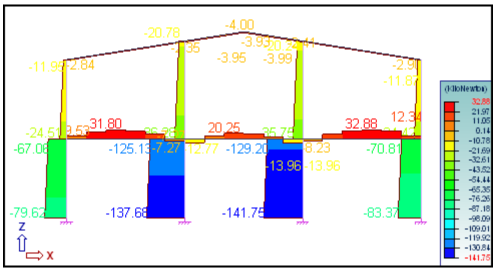


Figure 243: Values on diagrams for forces on linear elements along the local **x**-axis

8. On the **Analysis - F. E. Results** toolbar, click  to save the post-processing view.

The view is saved in the Pilot.

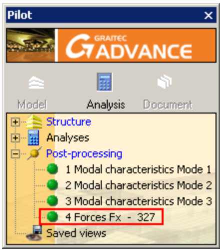


Figure 244: Forces on the **X**-axis is saved in the Pilot

Saving the post-processing view for forces along the local z-axis

1. On the **Analysis - F. E. Results** toolbar, make the following settings:
   - Result type: **Forces**.
   - Results on linear elements: **Fz** (shear force along the local **z**-axis).
   - Results on planar elements: **None**.
   - Select the **327** case combination.

Figure 245: **Analysis - F.E. Results** toolbar, analysis settings

2. Click  to start the post-processing.

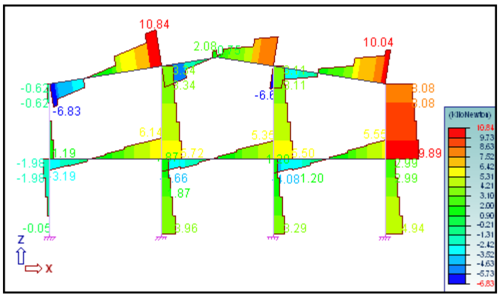


Figure 246: Forces along the local **z**-axis post-processing result

To select a specific color for the values on diagrams:

3. On the **Analysis - F. E. Results** toolbar, click .

4. In the "Results" dialog box, on the **Options** tab, in the "Values on diagrams" area, select **Display colors of values** option. The default color is black.

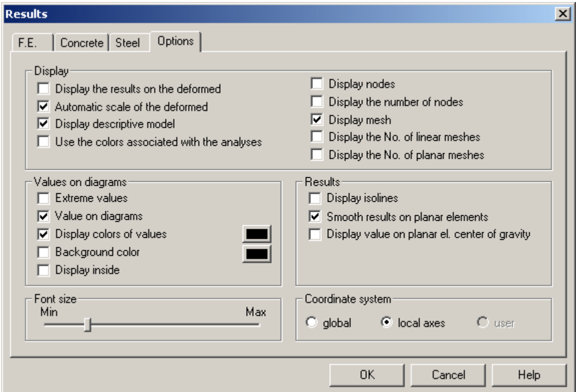


Figure 247: "Results" dialog box, display settings

5. Click **OK**.

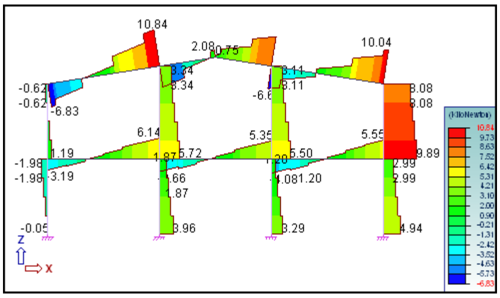


Figure 248: Display of the values on diagrams

Note: *To change the text color for the values display, click the color button next to the* ***Display color of values****.*

6. On the **Analysis - F. E. Results** toolbar, click  to save the post-processing view.

Saving the post-processing view for the bending moment about the local y-axis

1. On the **Analysis - F. E. Results** toolbar, make the following settings:

- Result type: **Forces**.
- Results on linear elements: **My** (bending moment about the local **y** axis).
- Results on planar elements: **None**.

– Select the **327** case combination.

Figure 249: **Analysis - F. E. Results** toolbar, analysis settings

2. Click to start the post-processing.

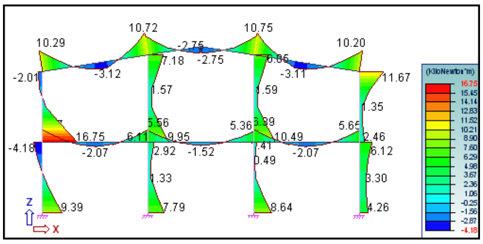


Figure 250: The bending moment results on the first portal frame

3. On the **Analysis - F. E. Results** toolbar, click to save the post-processing view.

Generating a report with efforts results on the first portal frame

1. Select the elements of the portal frame.

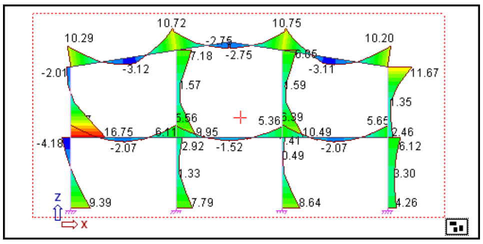


Figure 251: Selecting the first portal frame

Note: *The report generator considers the current selection. Therefore, the report content automatically refers only to the selected elements.*

2. From the main menu, select **Documents > Generate a new report**.

3. In the "Report generator" dialog box, click to empty the list.

4. On the **Table** tab, unwind **Finite Elements Analysis > Results > Forces > Linear elements** and double click **Linear elements forces by load case** to add it in the Document structure.

5. Select the table you just added and click **Table properties** to modify the contents of this table.

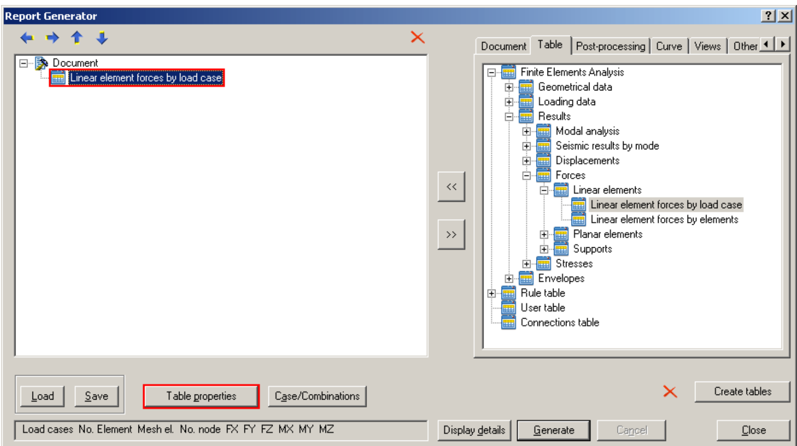


Figure 252: "Report generator" dialog box, adding **Linear elements forces by load case** it in the Document structure

6. In the “Table properties” dialog box, make the following settings:

– Click the **Advanced options** button to display the “Specific selection” panel.

– Select the **Define a specific selection of loads cases \ combinations** option.

– From the “Analysis type” drop-down list, select **None**.

– Select the **327** load combination.

– In the “Results option” area, from the “Results on” drop-down list, select **All quarters (super element)**.

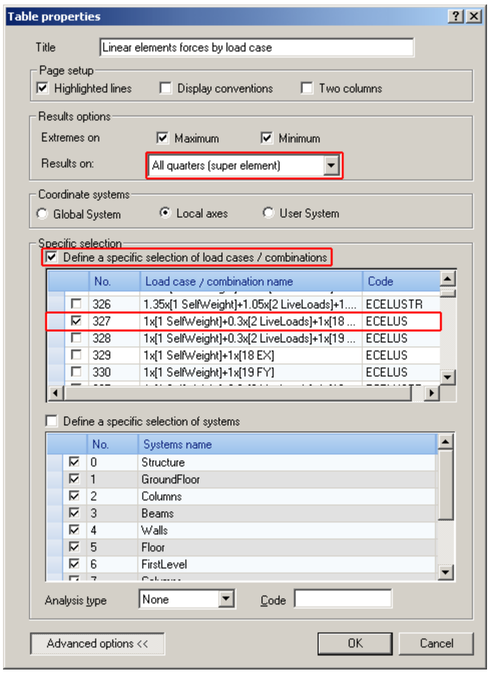


Figure 253: “Table properties” dialog box with “Specific selection” panel, setting the report contents

7. Click **OK**.

8. In the “Report generator” dialog box, on the **Post-processing** tab, insert the forces saved views in the Document structure.

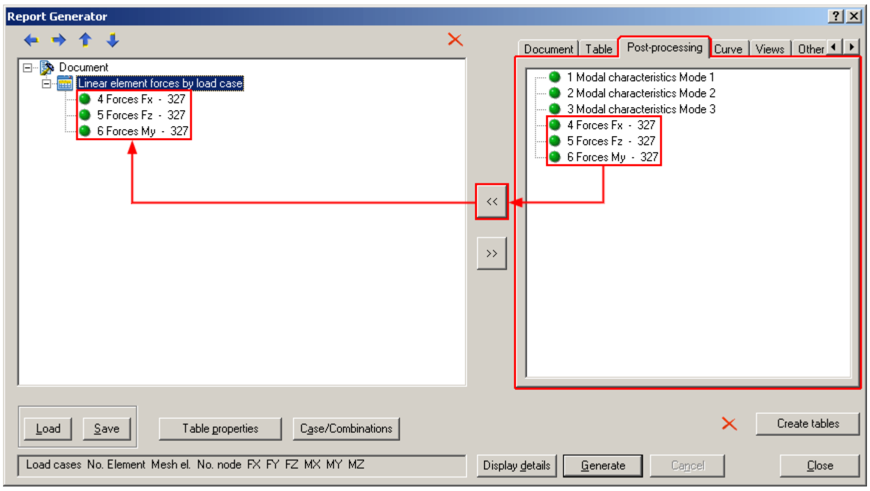


Figure 254: “Report generator" dialog box” inserting the new views

9. Select the **Table** tab, unwind **Finite Elements Analysis > Results > Forces > Supports > Point supports** and double click **Point supports actions by load case** to add it in the Document structure.

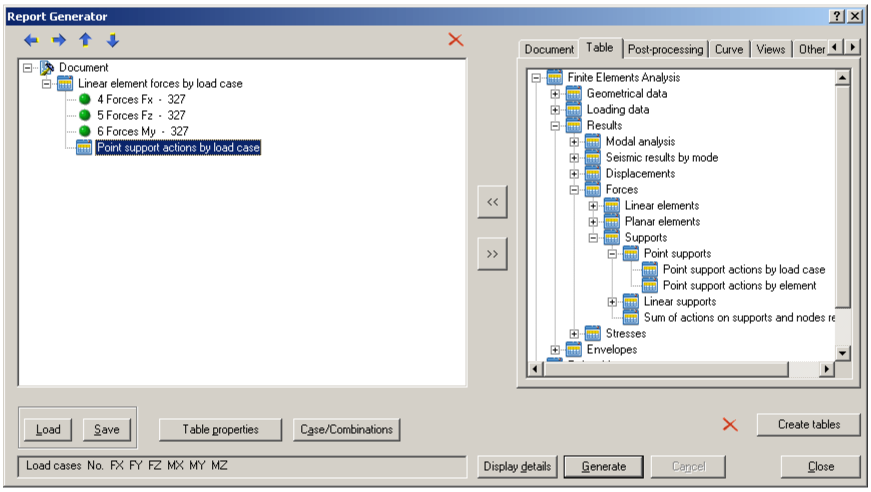
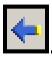


Figure 255: “Report generator” dialog box, adding the **Point supports actions by load case** in the Document structure

10. To set the position of this table in the Document structure, select it and click .

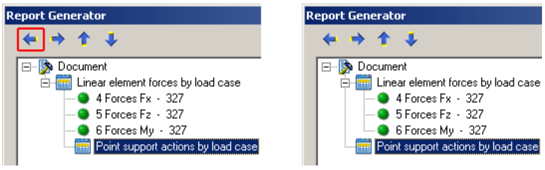


Figure 256: Changing the root of the **Point supports actions by load case**

11. Select **Point supports actions by load case** and click **Table properties**.
12. In the “Table properties” dialog box, click the **Advanced options** button.
13. On the “Specific selection” panel, make the following settings:
    - Select the **Define a specific selection of loads cases / combinations** option.
    - From the “Analysis type” drop-down list, select **None**.
    - Select the **327** load combination.
14. Click **OK**.
15. In the “Report generator” dialog box click **Generate**.

The .rtf report file opens. It contains the “Linear element forces by load case (local coordinate system)” table with the values calculated for the **327** combination case, the saved views and the “Point support actions by load case (global coordinate system)” table.

Lesson 12: Steel design and shape optimization

The "Steel Design Expert" allows the verification of the cross sections resistance, the elements stability and the optimization of the steel shapes. In this lesson you will use the steel design module to configure the design properties of steel elements.

You will learn how to:

- Launch the steel element analysis.
- View results on steel elements.
- Optimize the steel shapes.

Before starting:

On the **Filters and selection** toolbar, click  to display all the elements of the model.

Step 1: Define local assumptions for steel design

1. On the **Filters and selection** toolbar, from the "Selection by" drop-down list, select **Materials**.

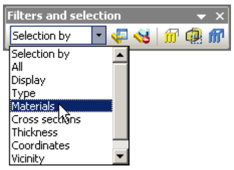


Figure 257: **Filters and selection** toolbar, selection by material

2. In the "Elements selection" dialog box, from the "Materials" list, select **S235** and click **OK**.

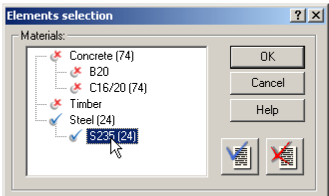


Figure 258: "Elements selection" dialog box, selecting the **S235** material

All the steel elements of the model are selected.

3. In the Properties window, make the following settings:

   Design experts

   – In the "Design experts" category, expand **Deflections** and enter **150** in the **Allowable deflection 1** field.

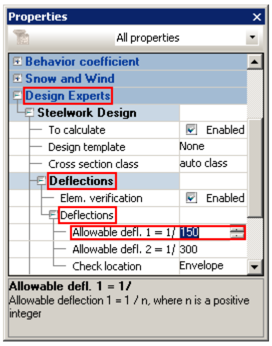


Figure 259: "Design experts" category, entering the **Allowable deflection 1** value

– In the “Buckling” category, expand **Buckling lengths**, click **Buckling lengths** , then click .

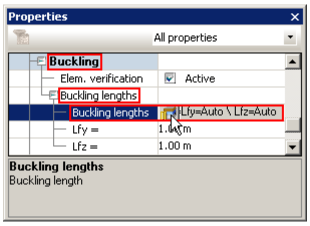


Figure 260: Accessing the “Buckling” dialog box

The “Buckling” dialog box appears. It allows the definition of buckling lengths for the selected elements.

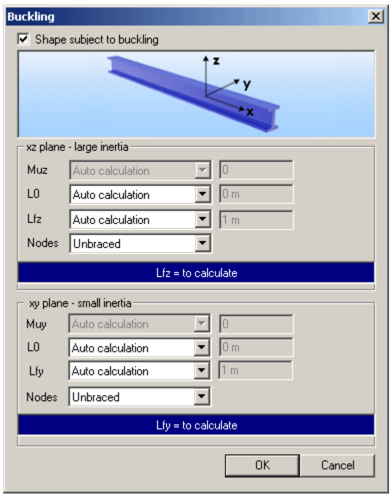


Figure 261: “Buckling” dialog box with default settings

The deviation coefficients from the critical stress **Muy** and **Muz** are calculated automatically or can be set by the user.

Click **OK** to close the “Buckling” dialog box.

– In the “Lateral-torsional buckling” category, disable the **Element verification** option.

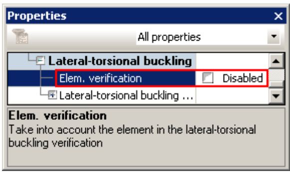


Figure 262: Disabling the **Element verification** option

Unselect all the elements of the structure by pressing <**Esc**>.

Step 2: Launch the steel calculation

From the main menu, select **Analyze > Steel calculation**. The calculation starts.

When the steel calculation is completed, the **Analysis - Steel Results** toolbar appears.

Figure 263: The **Analysis - Steel Results** toolbar

Step 3: View stability results on steel elements

Press <**ALT + 5**> for a 3D view.

1. On the **Analysis - Steel Results** toolbar, make the following settings:
   – Result type: **Stability**.
   – From the "Stability" drop-down list, select **Work ratio**.

Figure 264: **Analysis - Steel Results** toolbar, analysis settings

2. On the **Analysis - Steel Results** toolbar, click to start the post processing. The results are displayed in percentages.

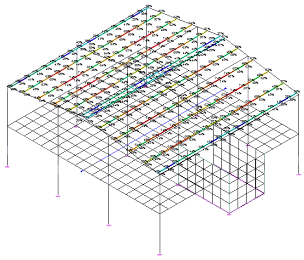
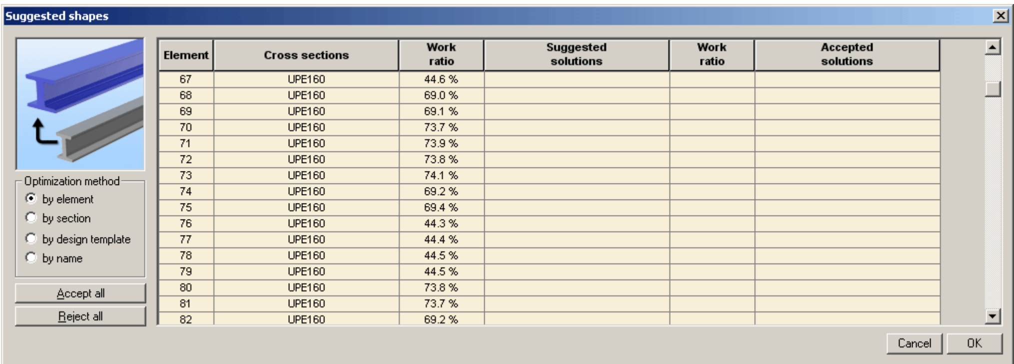


Figure 265: Stability on steel elements post-processing result

Step 4: Optimizing the steel shapes

The "Steel Design Expert" compares the work ratio of the steel elements and proposes other cross sections, that would correspond to the defined conditions. The suggested shapes can be globally or partially accepted. After accepting the suggested solutions, the model must be recalculated with the steel expert. These operations can be iterated until you obtain a work ratio comprised in the specified range for all steel shapes.

1. On the **Analysis - Steel Results** toolbar, click to access the "Suggested shapes" dialog box. If there are any suggestions, the table displays them in the "Suggested solutions" field.



Element	Cross sections	Work ratio	Suggested solutions	Work ratio	Accepted solutions
67	UPE160	44.6 %			
68	UPE160	69.0 %			
69	UPE160	69.1 %			
70	UPE160	73.7 %			
71	UPE160	73.9 %			
72	UPE160	73.8 %			
73	UPE160	74.1 %			
74	UPE160	69.2 %			
75	UPE160	69.4 %			
76	UPE160	44.3 %			
77	UPE160	44.4 %			
78	UPE160	44.5 %			
79	UPE160	44.5 %			
80	UPE160	73.8 %			
81	UPE160	73.7 %			
82	UPE160	69.2 %			

Figure 266: "Suggested shapes" dialog box

2. The optimization can be done by element, section, design template or name. Select the corresponding optimization method.
3. Click the "Accepted solutions" field and select the appropriate cross section from the drop-down list.
4. Click **OK** to apply.

If no other suggestions are displayed in the "Suggested shapes" dialog box, click **OK** to close the window.

Note: *After the optimization, re-launch the steel calculation. It is recommended to run the finite elements analysis together with the steel calculation.*

Lesson 13: Reinforced concrete design

The reinforced concrete design allows you to verify and calculate the reinforcement of concrete structure elements and to verify the concrete cross sections considering the action effects and the reinforcement solution.

You will learn how to:

- Launch the reinforced concrete analysis.
- View the reinforced concrete results.
- Generate reports of reinforcement results.

Step 1: Select combinations

In this step, select the combinations to use in the reinforced concrete calculation.

1. In the Pilot, right click **Combinations** and select **Properties** from the context menu.

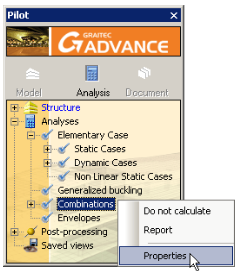


Figure 267: **Combinations** context menu

2. In the "Combinations" dialog box, select the **Concrete** tab and click **Modify list**.

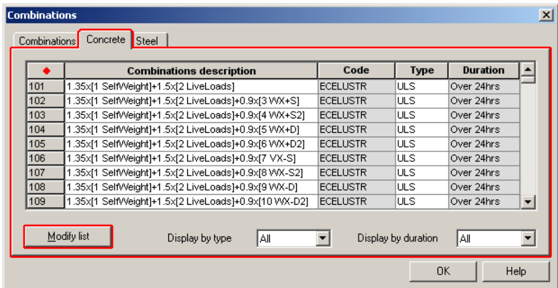


Figure 268: "Combinations" dialog box

3. By default, all the available combinations are calculated and displayed in the "Selected combinations" list. Select only the combinations between **101** and **105**:

   – In the "Selected combinations" area, click the **All** button to select the combinations, then click  to remove them.

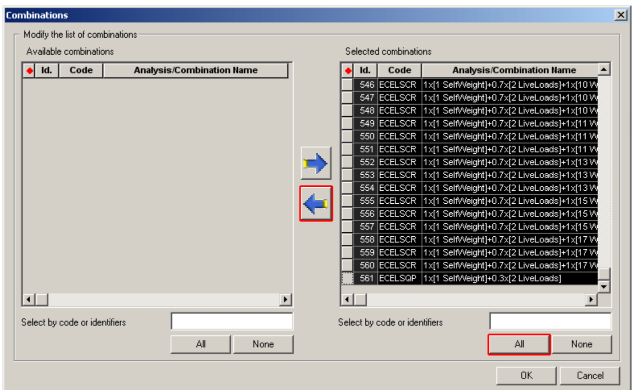


Figure 269: Removing the combinations

– In the “Available combinations” list, select combinations **101** - **105** and click to add them in the “Selected combinations” list.

*Click the first item to select, keep the <**SHIFT**> key pressed and click the last item to select consecutive items faster.*

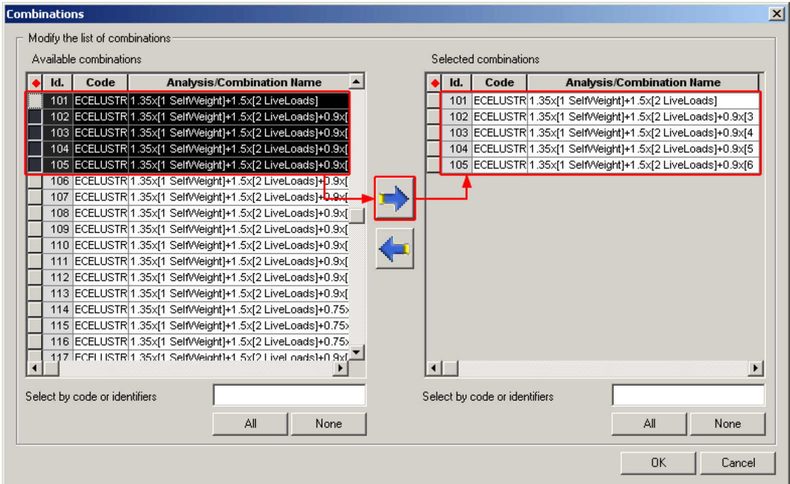


Figure 270: Adding the specified combinations in the “Selected combinations” list

4. Click **OK**.

Display all the model elements:

On the **Filters and selection** toolbar, click .

Step 2: Launch the reinforced concrete analysis

From the main menu, select **Analyze > Reinforced Concrete calculation**. When the reinforced concrete calculation is completed, the **Analysis - Reinforced Concrete Results** toolbar appears.

Figure 271: The **Analysis - Reinforced Concrete Results** toolbar

Step 3: View the reinforced concrete post-processing results

You can select different visualization modes of Concrete Design results in the graphic area.

Viewing the reinforced concrete results for the linear elements along the Z-axis

Press <**ALT + 2**> for a right view.

1. Select the first portal frame.

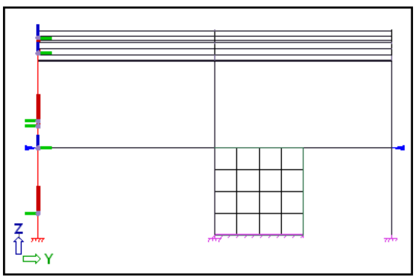


Figure 272: Selecting the elements of a portal frame

2. On the **Filters and selection** toolbar, click  to display only the selected elements.

Press <**ALT + 1**> for a front view.

3. On the **Analysis - Reinforced Concrete Results** toolbar, make the following settings:
   – Result type: **Reinforcement**.
   – Results on linear elements: **Az** (longitudinal reinforcement area along the **z**-axis).
   – Results on planar elements: **None**.

Figure 273: The **Analysis - Reinforced Concrete Results** toolbar, analysis settings

4. Click  to start the post-processing.

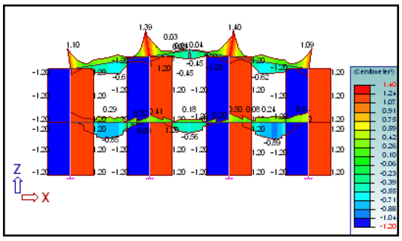


Figure 274: Reinforced concrete post-processing results for the linear elements along the **Z**-axis

Saving the post-processing view

On the **Analysis - Reinforced Concrete Results** toolbar, click  to save the post-processing view.

The view is saved in the Pilot, in **Post-processing**.

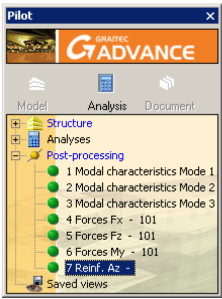


Figure 275: The post-processing view, in the Pilot

Saving the result curves on a beam

1. Keep the <**Esc**> key pressed to clear the previous results display, and then select a beam of **FirstLevel**.

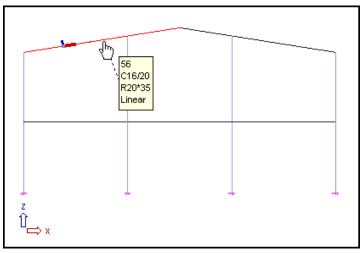
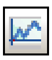
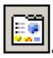


Figure 276: Selecting a beam

2. On the **Analysis - Reinforced Concrete Results** toolbar, click 

3. In the “Result curves” dialog box, click .

4. In the “Curves” dialog box, on the **Concrete** tab, select **Linear element** and the **Ahz** option.

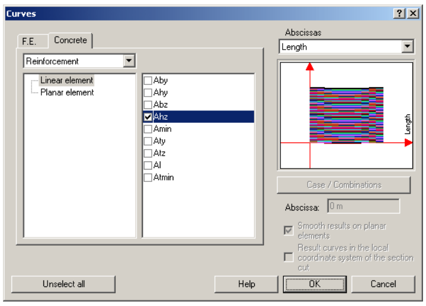
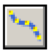


Figure 277: “Curves” dialog box, results settings

5. Click **OK** to apply and close the dialog box.

6. Double click in the graphic area.

7. In the “Curves” dialog box, click  to display the intersection points.

8. Click  to save the reinforcement diagram for use in the reports.

9. In the “Result curves” dialog box enter **Reinforcement Ahz** for the diagram name.

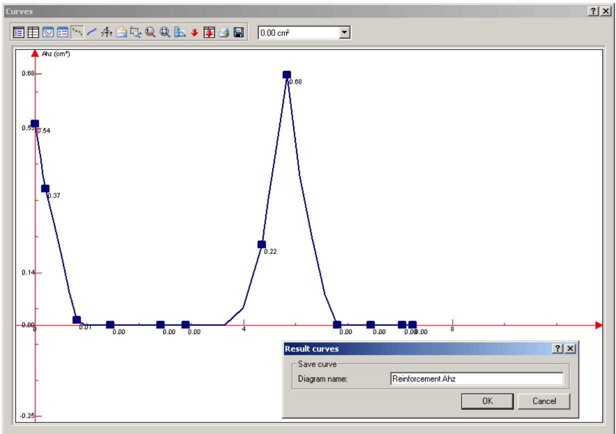


Figure 278: “Result curves” dialog box, naming the diagram

Using the same process, save the post-processing results for Abz:

1. In the “Result curves” dialog box make the following settings:

– Result type: **Reinforcement**.

– From “Reinforcement” drop-down list, select **Abz**.

2. Click  to start the post-processing.

3. Double click in the graphic area.

4. In the “Curves” dialog box, click  to save the reinforcement diagram for use in the reports.

5. In the “Diagram name” field enter **Reinforcement Abz**.

Saving the post-processing view of the longitudinal upper and lower reinforcement area along the local z-axis

1. On **Analysis - Reinforced Concrete Results** toolbar, click .
2. In the “Results” dialog box, **Concrete** tab, select the **Linear** and **Az** options.
3. Click **OK**.

The post-processing starts.

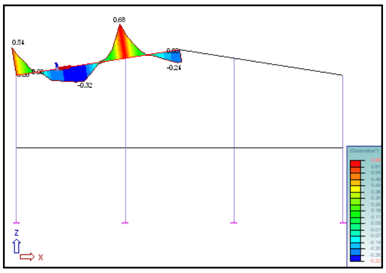


Figure 279: Result curves on a beam

4. On **Analysis - Reinforced Concrete Results** toolbar, click to save the post-processing view.

Step 4: Generate the report

1. From the main menu, select **Documents > Generate a new report**. The “Report generator” dialog box appears. Click to empty the Document list.
2. On the **Table** tab, expand **Reinforced Concrete Analysis > Results > Reinforcement areas linear elements** and double click **Longitudinal reinforcement linear elements** to add it in the Document structure.
3. On the **Post-processing** tab, double click the new views, **7 Reinf. Az** and **8 Reinf. Az**, to add them in the Document structure.
4. On the **Curve** tab, double click the **Abz** and **Ahz** views to add them in the Document structure.

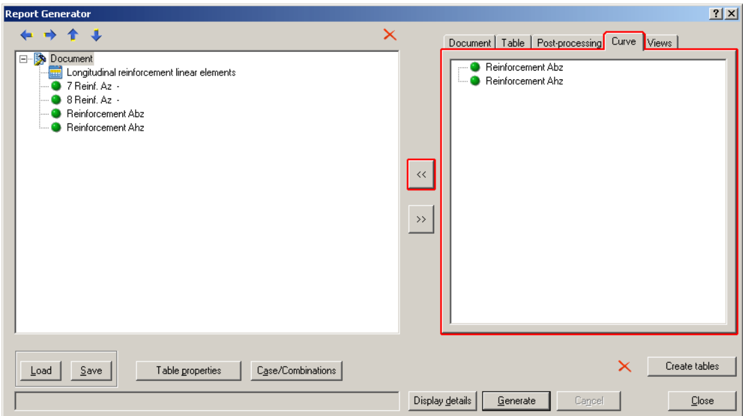


Figure 280: Adding the views in the Document list

5. Click **Generate**.

The .rtf report file opens. It contains the “Linear element longitudinal reinforcement (local coordinate system)” table, the reinforcement views and the curves diagrams.

Viewing the floor inferior reinforcement along the X-axis

Before starting:

On the **Filters and selection** toolbar, click .

Press <**ALT + R**>and rotate for a 3D view.

1. Select the floor.

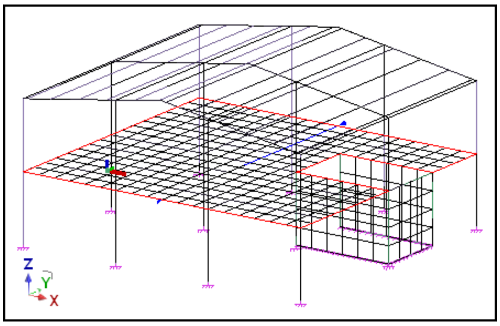


Figure 281: Selecting the floor

On the **Filters and selection** toolbar, click to display only the selected element.

Press <**ALT + 3**> for a top view.

Double click the scroll mouse button for a fit on screen view of the element.

2. On the **Analysis - Reinforced Concrete Results** toolbar, make the following settings:
   - Result type: **Reinforcement**.
   - From the results on linear elements drop-down list, select **None**.
   - From the results on planar elements drop-down list, select **Axi** (inferior reinforcement along the **X**-axis).

Figure 282: The **Analysis - Reinforced Concrete Results** toolbar, analysis settings

3. Click to start the post-processing.

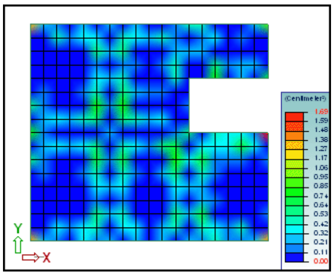


Figure 283: Inferior reinforcement along the **X**-axis on the floor

Viewing the results as histograms

Next, you will view the results in a non-linear distribution of color regions.

1. On the **Analysis - Reinforced Concrete Results** toolbar, click .

2. In the “Color table” dialog box, click for a non-linear distribution of color regions.

3. Click **Apply**. Click **OK** to close the dialog box.

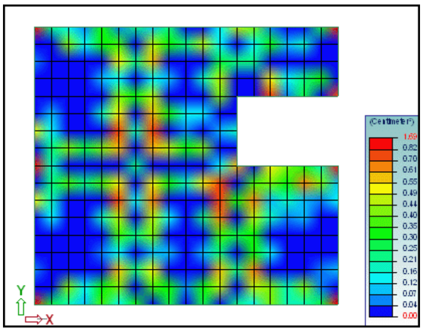


Figure 284: Inferior reinforcement along the **X**-axis post-processing result viewed as histograms

Viewing the result for the inferior reinforcement along the Y-axis on the floor

1. On the **Analysis - Reinforced Concrete Results** toolbar, make the following settings:
   - Result type: **Reinforcement**.
   - Results on linear elements: **None**.
   - Results on planar elements: **Ayi** (inferior reinforcement along the **y**-axis).

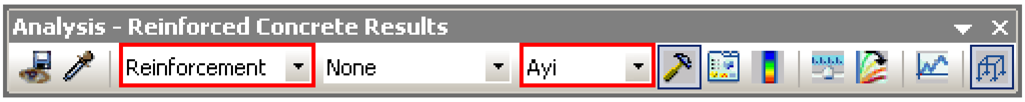


Figure 285: The **Analysis - Reinforced Concrete Results** toolbar, analysis settings

2. Click  to start the post-processing.

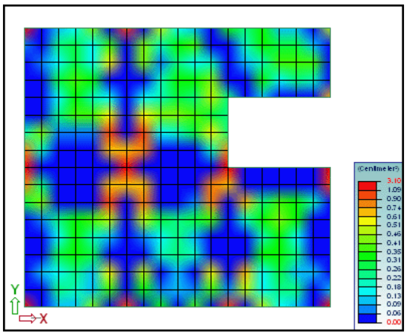


Figure 286: Inferior reinforcement along the **y**-axis on the floor

Viewing the result for the superior reinforcement along the x-axis on the floor

1. On the **Analysis - Reinforced Concrete Results** toolbar, make the following settings:
   - Result type: **Reinforcement**.
   - Results on linear elements: **None**.
   - Results on planar elements: **Axs** (superior reinforcement along the **x**-axis).

Figure 287: The **Analysis - Reinforced Concrete Results** toolbar, analysis settings

2. Click  to start the post-processing.

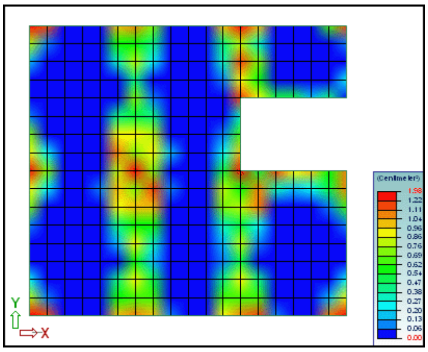


Figure 288: Superior reinforcement along the local **x**-axis on the floor

Viewing the result for superior reinforcement along the y-axis on the floor

1. On the **Analysis - Reinforced Concrete Results** toolbar, make the following settings:

– Result type: **Reinforcement**.

– Results on linear elements: **None**.

– Results on planar elements: **Ays** (superior reinforcement along the **y**-axis).

Figure 289: The **Analysis - Reinforced Concrete Results** toolbar, analysis settings

2. Click  to start the post-processing.

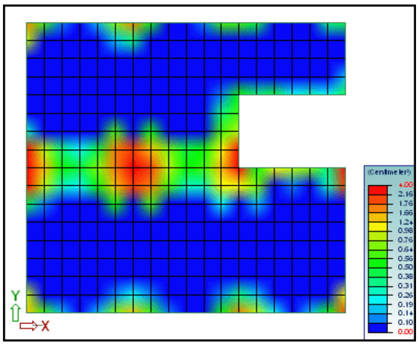


Figure 290: Superior reinforcement along the **y**-axis on the floor

Viewing the result curves on the section cut

<u>Before starting</u>:

On the **Filters and selection** toolbar, click .

Press <**ALT + R**> and rotate for a 3D view.

1. Select the section cut.

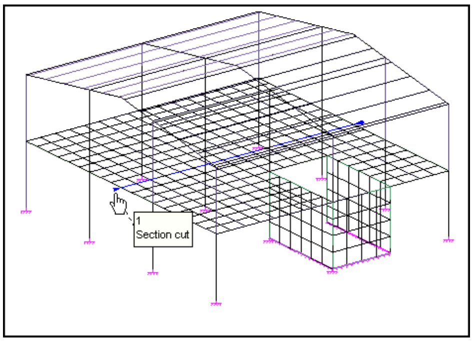
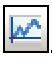


Figure 291: Selecting the section cut

2. On the **Analysis - Reinforced Concrete Results** toolbar, click .

3. On the “Result curves” dialog box, click  to set the results options.

4. In the "Curves" dialog box, on the **Concrete** tab, make the following settings:
   – Select results on **Planar elements**.
   – Select **Axi** to see inferior reinforcement along the local **x**-axis.
   – Select **Ayi** to see inferior reinforcement along the local **y**-axis.

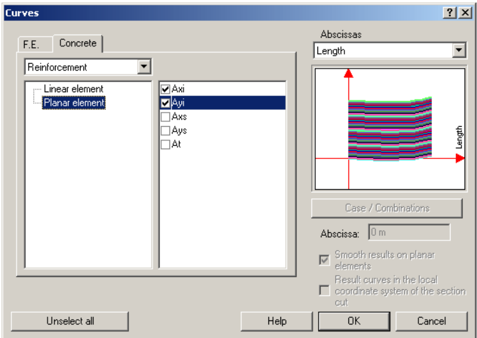
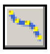


Figure 292: “Result curves” dialog box

5. Click **OK** to apply.
   – Double click the **Axi** graphic. The “Curves” dialog box appears. Click  to display the curve points.

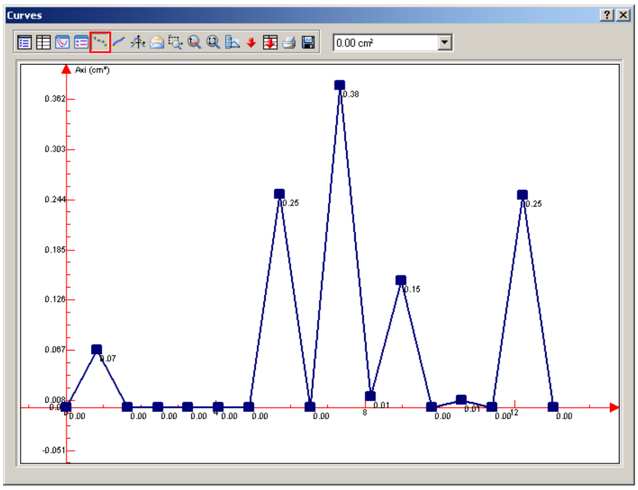
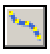


Figure 293: Inferior reinforcement along the local **x**-axis

   – Double click the **Ayi** graphic. The “Curves” dialog box appears. Click  to display the curve points.

Lesson 14: Column reinforcement analysis

In this lesson you will use the column reinforcement analysis tool.

You will learn how to:

- Modify the calculated reinforcement for a column.
- Verify the column reinforcement with interaction curves.

Viewing the reinforcement concrete results for the elements of the first portal frame

Press <**ALT + 2**> for a right view.

1. Select the first portal frame.

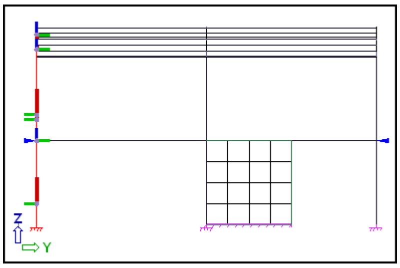


Figure 294: Selecting the first portal frame of the structure

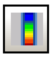
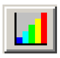
On the **Filters and selection** toolbar, click .

Press <**ALT + 1**> for a front view.

2. On the **Analysis - Reinforced Concrete Results** toolbar, make the following settings:
   - Result type: **Reinforcement**.
   - Results on linear elements: **Az** (longitudinal reinforcement area along the **z**-axis).
   - Results on planar elements: **None**.

Figure 295: **Analysis - Reinforced Concrete Results** toolbar, analysis settings

3. Click  to start the post-processing.

4. Click  to access the “Color table” dialog box and click  to select a linear distribution of results.

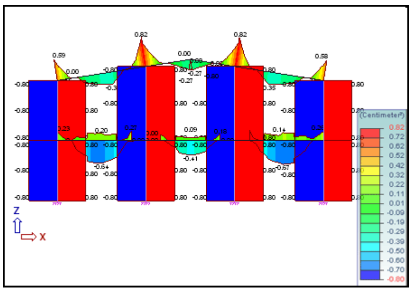


Figure 296: Reinforcement concrete results for the elements of the first portal frame

Viewing the column reinforcement

During the concrete design calculation, the concrete expert determines the appropriate reinforcement for each column and suggests the reinforcement parameters.

1. Select the second column of **GroundFloor**. (Press <**Esc**> to clear previous results).

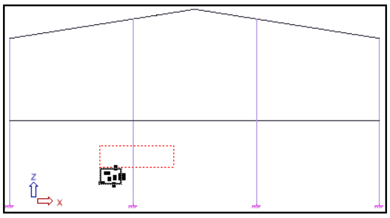


Figure 297: Selecting a column of the **GroundFloor** system

2. Access the “Modifications of longitudinal reinforcements bars” dialog box:
   - In the Properties window, go to the “Design experts” category.
   - Expand **Column calculation > Reinforcement**, click **Reinforcement**, and then click .

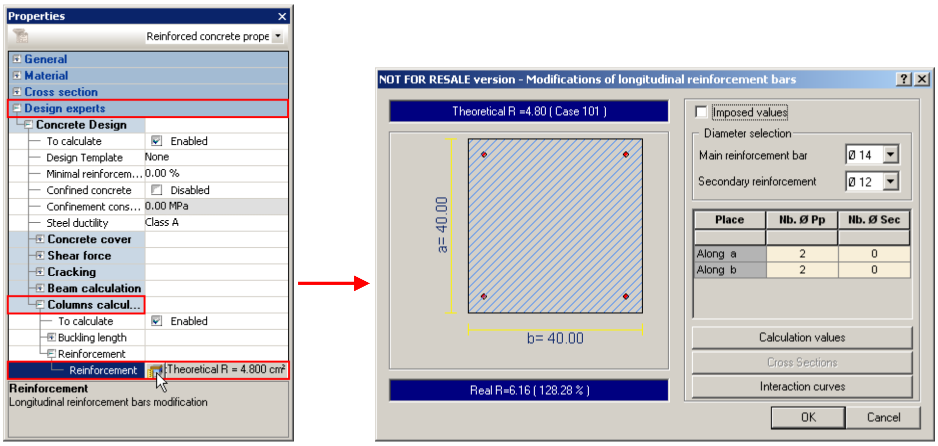


Figure 298: Accessing the “Modifications of longitudinal reinforcements bars” dialog box

The “Modifications of longitudinal reinforcements bars” dialog box displays the calculated reinforcement determined by the concrete expert and allows the user to modify the reinforcement parameters.

Modifying the reinforcement parameters

The “Theoretical R” field is highlighted in red when the defined diameter or the defined number of bars make the real reinforcement area smaller than the requested value.

1. In the “Diameter selection” area modify the diameter of the main and secondary reinforcement bars by selecting the appropriate diameter from the drop-down list.
2. To add /remove main or secondary reinforcement bars, enter the desired numbers in the table.

   Notice the reinforcement graphic on the left side of the “Modifications of longitudinal reinforcements bars” dialog box. It displays the configuration of the reinforcement bars and the size of their diameter.
3. To consider the user-defined values when recalculating the model, enable the **Imposed values** option.

View the interaction curves for the Mx and My bending moments

1. In the “Modifications of longitudinal reinforcements bars” dialog box, click **Interaction curves**. The “Interaction curves” dialog box appears.

2. From the drop-down list, select **My/Mz**.

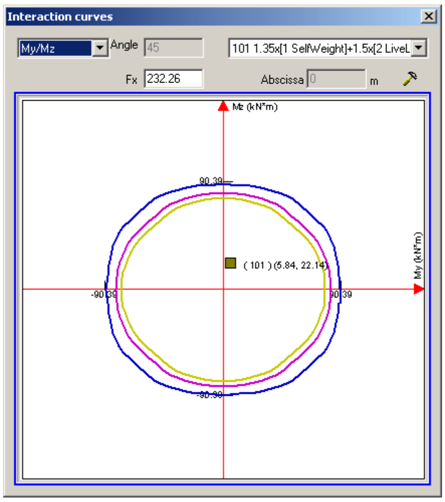


Figure 299: “Interaction curves” dialog box

3. Double click in the graphic area. The “Curves” dialog box appears.

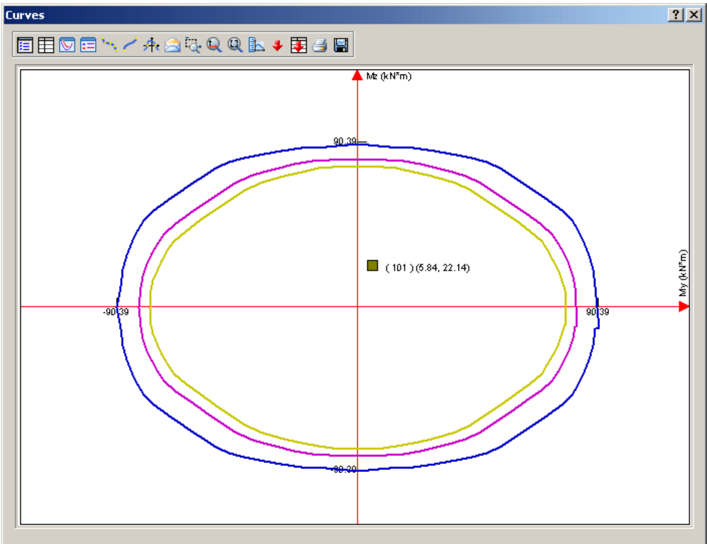


Figure 300: “Curves” dialog box

The interaction curves for the **Mx** and **My** bending moments are displayed. Close the “Curves” dialog box.

Viewing the classical axial force / moment interaction curves on My

1. On the “Interaction curves” dialog box, make the following settings:
   – Result type: **Fx/My**.
   – Select the **101** case combination.
   – (The angle is 45°).
2. Click to start the post-processing.
3. Double click in the graphic area. The “Curves” dialog box appears:

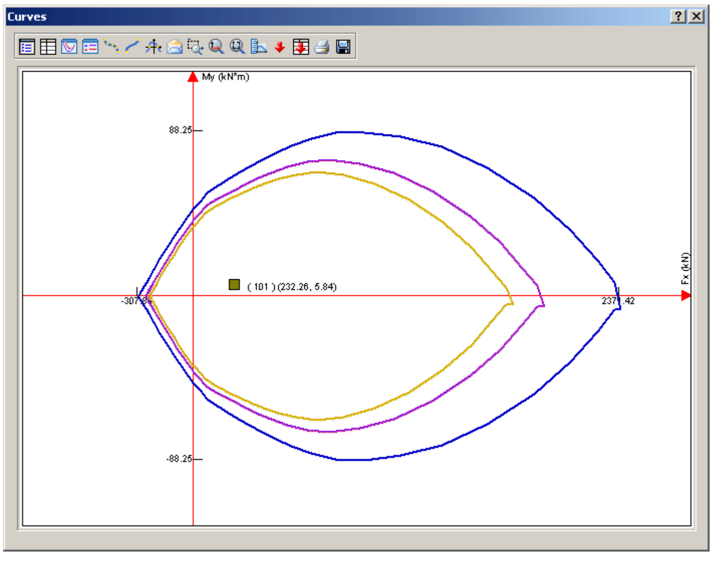


Figure 301: “Curves” dialog box

Viewing the axial force / bending moment interaction curves on Mz

1. In the “Interaction curves” dialog box, from the drop-down list, select **Fx/Mz**.
2. Click to start the post-processing.
3. Double click in the graphic area. The “Curves” dialog box appears.

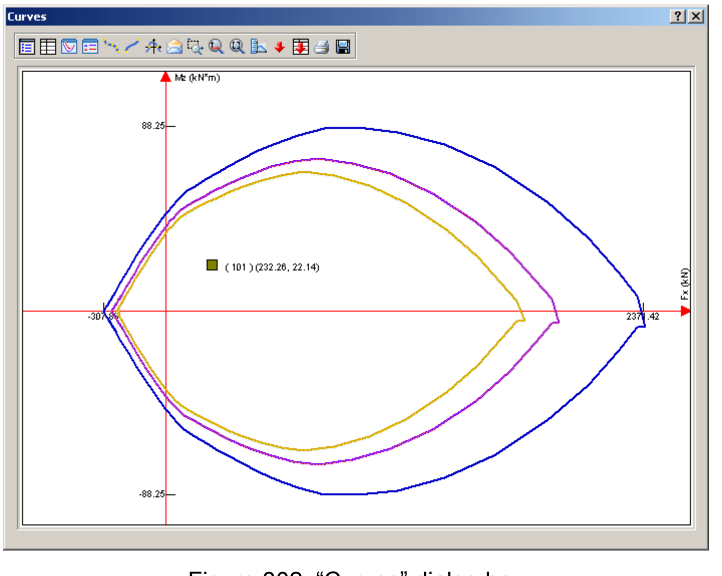


Figure 302: “Curves” dialog box

Lesson 15: Creating a post-processing animation

With Advance Design you can create an animation starting from different views of the model. The animation represents the transition between cameras by a specified frame rate.

You will learn how to:

- Create camera views around the model.
- Launch the post-processing animation.

Step 1: Display the eigen mode 2 result

On the **Filters and selection** toolbar, click .

Press <**ALT + 5**> for a 3D view.

1. On the **Analysis - F.E. Results** toolbar, make the following settings:
   – Result type: **Eigen modes**.
   – From the list of **Eigen modes** analysis, select **Eigen mode 2**.

Figure 303: **Analysis - F. E. Results** toolbar

2. Click  to start the post-processing.

Step 2: Add cameras around the model

1. On the **Animation** toolbar, click . A camera is placed on the center of the current view.
2. Press <**ALT + R**> and drag to change the view angle.

Using the same process, create several cameras.

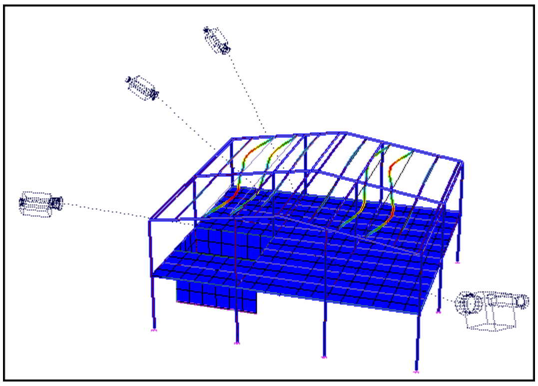


Figure 304: Placing cameras around the model

Hide the cameras: on the **Animation** toolbar, click .

Step 3: Launch the animation

Next, view the animation as a translation between cameras:

1. On the **Animation** toolbar, click .

2. In the “Animation options” dialog box, on the **General options** tab, select the **Transitions between cameras** option.

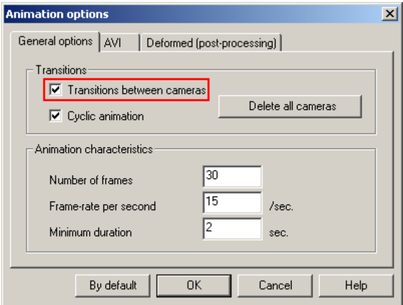
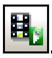


Figure 305: “Animation options” dialog box

3. Click **OK** to apply.

4. On the **Animation** toolbar, click .

The animation is launched in the graphic area.

5. Press <**Esc**> to end the animation.

ADVANCE DESIGN

GRAITEC

www.graitec.com

031514-0409S-0418